Performance Analysis of Vertical Axis Wind Turbine with Comparison of CFD and Experimental Analysis

Mayank D. Patel
M.E. Student
Department of Mechanical Engineering
Government Engineering College, Dahod-391351

Tushar P. Gundarneeya
Assistant Professor
Department of Mechanical Engineering
Government Engineering College, Dahod-391351

Abstract

Computational fluid dynamics (CFD) simulations were performed in the present study using ANSYS CFX, a commercially-available CFD package, to characterize the behaviour of the new VAWT. Static three-dimensional CFD simulations were conducted. The static torque characteristics of the turbine and the simplicity of design highlight its suitability for the small wind turbine market. The major factor for generating the power through the VAWT is the velocity of air and the position of the blade angle in the VAWT blade assembly. The study presents the effect of the position of the blade angle from 0 degree to the 30 degree on the power developed by the VAWT. The experimental analysis will be conducted in wind tunnel so that the result of both CFD analysis and experimental analysis can be compared. CFD workbench of ANSYS is used to carry out the virtue simulation and testing. The software generated test results are validated through the experimental readings. Through this obtainable result will be in the means of maximum constant power generation from VAWT.

Keywords: CFD, Vertical Axis, Energy.

I. INTRODUCTION

The focus on Renewable Energy Resources has increased significantly in the recent years in the wake of growing environmental pollution, rising energy demand and depleting fossil fuel resources. Different sources of renewable energy include biomass, solar, geothermal, hydroelectric, and wind. Among these resources wind has proved to be a cheaper alternative energy resource and hence extensive research efforts have been put to improve the technology of electricity generation through wind. The world has enormous potential of wind energy that can be utilized for electricity generation. Currently, large scale VAWTs are not economically attractive; however, they offer energy solutions for remote places away from the main distribution lines and places where large wind farms cannot be installed due to environmental concerns and small scale dispersed generation units are preferred. That is why mass production of VAWTs has recently been started as small scale wind power generating units.

II. CONFIGURATIONS

A great degree of design versatility exists in the vertical axis wind turbines as shown in Table. There are a few problems inherent to the currently available designs including low starting torque, blade lift forces, low efficiency, poor building integration, etc. In the past few decades, the engineers came up with many new and innovative design approaches to resolve these issues associated with VAWTs. A detailed review of VAWT configurations available till now and the work done on each configuration has been discussed in the following sections. Darrieus wind turbine designs were first patented in 1931. These types of turbines have highest values of efficiency among VAWTs but generally suffer from problems of low starting torque and poor building integration. Darrieus type wind turbines have many variants, as discussed in the following lines, all of which are Lift-type wind turbines, i.e. lift forces acting on the blades of turbine cause the rotor to rotate and hence generate electricity.

III. LITERATURE SURVEY

Vertical axis wind turbine offer economically viable energy solution for remote areas away from the integrated grid systems. In order to spread the use of VAWT, the problems associated with various configurations, i.e. poor self-starting and low initial torque, low coefficient of power, poor building integration should be overcome. Furthermore, following conclusions can be drawn from the present review:

- Ample wind energy potential is available in the world. In order to make best use of it efficient designs of wind turbines need to be developed.
- Various vertical axis wind turbines can offer solution to the energy requirements ranging from 2 kW to 4 MW with a reasonable payback period.

- Coefficient of power can be maximized by selecting a suitable TSR range for various configurations.
- VAWTs offer good possibility of building integrated designs. Crossflex type VAWT can be used on high rise buildings in the cities where free stream velocity greater than 14 m/s is available. Similarly, Sistan type wind turbine can be effectively integrated with building designs and can give reasonable power output.
- CFD is capable of designing the VAWT with higher degree of accuracy. It can also be used for the optimization of blade design. Moreover, flow field around various configurations' blades can also be visualized with the help of CFD. It has not only accelerated the design process of VAWT but also has brought down the overall cost of designing.
- Calculation of Energy efficiency can give a better understand-ing of the losses occurring in the VAWTs due to irreversibility's. Targeted efforts can then be made to overcome these losses.

The straight turbine rotor blade, with an aspect ratio of 4:1, operates at relatively low tip speeds and its performance shows a clear dependence on the rotor blade surface finish. Below a critical Reynolds number (30,000), the performance is enhanced by having the surface of the turbine roughened, but above this Reynolds number the power coefficient is degraded. The tests also shows that the two and three bladed rotor models produces similar peaks in performance coefficient, but that the three bladed design did so at a much reduced TSR.

- Computational predictions of the performance coefficient of this turbine were carried out and the 3D simulations were shown to be in reasonably good agreement with the experimental measurements, considering errors and uncertainties in both the CFD simulations and the wind tunnel measurements. The 2D simulations showed a significantly increased performance compared to the 3D simulations and this was shown to be mainly due to the presence of the large tip vortices present in the real turbine and the 3D simulations. Simulations illustrated the periodic pulsing nature of the tip vortices caused by the changing lift generated by the rotor blades as they travel through each rotor revolution. At phases where higher amounts of lift are generated, stronger tip vortices are present, whereas at phases where little lift is generated, the vortices are significantly reduced.

An assessment of the performance of a novel vertical axis wind turbine was conducted using RANS CFD simulations. Steady and rotating validation studies were conducted using experimental data for a Savonius rotor. The static and dynamic torque curves for the Savonius rotor were well characterised. Steady two-dimensional CFD simulations were conducted and it was determined that the Aeolun Harvester_ produces a comparable amount of static torque to existing Savonius rotors. Rotating three-dimensional CFD simulations demonstrated that the average dynamic torque generated by this new rotor decays more rapidly with increasing tip speed ratio than the torque output of existing Savonius rotors. Pressure coefficient contours and the local torque coefficient indicate that this rapid decay in torque with increasing tip speed ratio occurs due to stagnation effects acting on the convex side of the outer wall as the blade is retreating. These stagnation effects increase with increasing tip speed ratio and have similar characteristics to the torque production mechanisms observed on Savonius rotors. The verification, validation, and prediction presented in this work demonstrated the applicability of RANS in the development of vertical axis wind turbines. The results of the study, however, have shown that the shape of the inner and outer rotor walls should be the focus of future work as a means to increase the torque generated by the rotor to render it more competitive with existing designs.

- A numerical model (based on the application to CFD of the aerodynamic principles which are currently applied to BE-M theory for rotor performance prediction) for the evaluation of energy performance and aerodynamic forces acting on a straight bladed vertical-axis Darrieus wind turbine has been presented.
- A simplified aerodynamic model (based on the analysis of kinematic and dynamic quantities at discrete and fixed rotor azimuthal positions along blades trajectory) was also presented, allowing the correlation between flow geometric characteristics (such as blade angles of attack) and dynamic quantities (such as rotor torque and blade tangential and normal forces).
- The results of a 2-D full campaign of simulations were proposed for a classical NACA 0021 three-bladed rotor. Through the analysis of the distribution of the instantaneous torque coefficients and of the relative angles of attack as a function of azimuthal position for a single rotor blade, flow field characteristics were investigated for several values of tip speed ratio, allowing a comparison between rotor operation at optimum and lower Cp values.
- The obtained results have shown the reduction of blade relative angles of attack passing from lower to higher TSR values, due to the increasing influence of blade translational speed in the near-blade flow field. It has also been shown that the maximum torque values are generated during the upwind revolution of the turbine and for azimuthal positions where rotor blades are experiencing very high relative angles of attack, even beyond the stall limit. The azimuthal positions of maximum power extraction along blade trajectory have been located inside the 4th and 5th octants, probably due to the combination of a great energy extraction exerted by the rotor blade (due to the upwind operation of the rotor blade itself) and a relative high lever arm with respect to the rotor axis. The distribution of aerodynamic forces as a function of rotor blade azimuthal position should be further investigated.

The average rotor power coefficient is rather lower, the instantaneous power coefficient locally exceeds the Betz's limit three times per rotor revolution. This phenomenon, probably caused by a sudden pressure coefficient drop concerning the whole rotor disc and the surrounding flow region (thus violating the assumptions of Rankine-Froude actuator disc theory) should be further investigated.

A wide-ranging analysis was carried out to evaluate the energetic suitability of a Darrieus VAWT installation in the rooftop of a building in a reference European city. With this goal in mind, the first step of the analysis consisted on a numerical CFD analysis to characterize the flow field in the rooftop area of two buildings with different proportions with respect to both the average surrounding buildings height and their upwind building; in addition, the application of either a plan or a sloping roof was considered. The flow velocity modulus and direction (skew angle) were calculated for different oncoming wind profiles and compared to their level in the undisturbed wind. Under the assumption of Rayleigh distribution of the blowing wind, the results were projected into net available wind distributions in the rooftop of each building. As a second step, a numerical model was developed to account for the effects of the skew angle of the flow on the power performance of the H-Darrieus turbine: the modified turbine power curves were hence evaluated with the developed model with the flow conditions previously calculated for the study configurations.

- Finally, the results of the CFD simulations and the new turbine model were combined in a comparative feasibility analysis of a medium-size H-Darrieus turbine in the built environment.
- The analysis showed that notable increments (up to 70%) of the attended capacity factor in the rooftop area of an installation building in the urban environment can be achieved whenever a building reasonably higher than the average of the surrounding constructions is selected and suitable geometric proportions of the building itself with respect to its upwind building are fulfilled; otherwise, a constant detriment of the energy potential was noticed.
- In addition, a positive influence on the velocity increment in the rooftop area of the sloping angle of the roof was appreciated; this effect was maximized by the application of a sloping roof with an inclination angle of 8_, which was deemed to guarantee the more effective guidance to the flow which overcomes the building.
- Focusing on the application of a Darrieus turbine, it is also worth noticing that the skew angles attended in the rooftop of a building in a urban environment (15–35_), seem to ensure a further increase (up to 12%) of the attended energy harvesting thanks to the improved behaviour of these machines in skewed-flow conditions; on the basis of a specific model that was developed to account for this effect in the performance estimations of Darrieus VAWTs, this contribution is deemed to be maximized whenever a skew angle of approximately 25_ is achieved.

The pitch angle is an important parameter on the Cp, Ct and k (the maximum values obtained at about 10_).

- The chord length has a significant effect on the Cp, Ct and k (increasing the chord length increasing these values).
- The turbine radius has a remarkable effect on the Cp, Ct and k (increasing turbine radius increases these values in the range of this study).
- There is noticeable increase in Cp, Ct and k resulting from increasing the number of blades from two to four blades. Less increase in these values results from the increase in the number of blades from three to four blades.
- Symmetrical aerofoil (NACA 0024) results in higher Cp, Ct and k, and there is a little increase in these values resulting in changing aerofoil type from NACA 4520 to NACA 4420.[6]

CFD is used to predict the optimal angles of attack that produce maximum power outputs for an untwisted horizontal axis wind turbine for four wind speed cases. By using the 80% span as the basis for design, the finding indicates that the optimal angles of attack are the ones near the maximum lift point. The angles are slightly larger as the speeds are higher and this is consistent with the shift of the curves as the Reynolds numbers are increased. Under typical design conditions lift to drag ratio was proved theoretically and confirmed by the computation as insignificant design parameter.

IV. OBJECTIVES

A. OBJECTIVE OF WORK:-

- Create the 3-D model of the blade profile and blade for the simulation in SOLID-WORK.
- Create the mesh model of the blade.
- Create the cavity model for the cavity analysis for the CFD analysis and simulation.
- Manufacture the prototype blade with calculated profile parameter for the experimental analysis.
- Experimental analysis will be done in the wind tunnel.
- Comparison of the software simulation data and experimental analysis Result will be done.
- Validation of the result will be done by comparing the Experimental result with ANSYS result.

V. CFD ANALYSIS OF VERTICAL AXIS WIND TURBINE BLADE

A. First Create Cavity of the VAWT in Solid works

CFX Analysis is performed by Cavity Pattern Analysis method. So we have to Create Cavity of the VAWT in which Air flows.

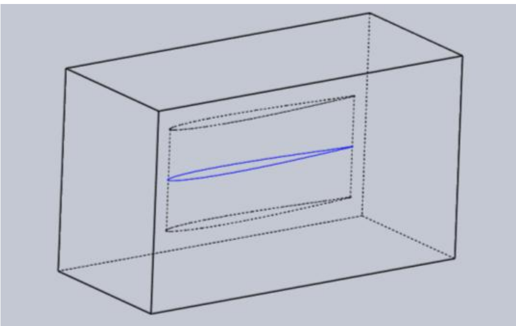


Fig. 1: Cavity of VAWT

B. Import IGES model of Air Cavity in ANSYS Workbench for Meshing

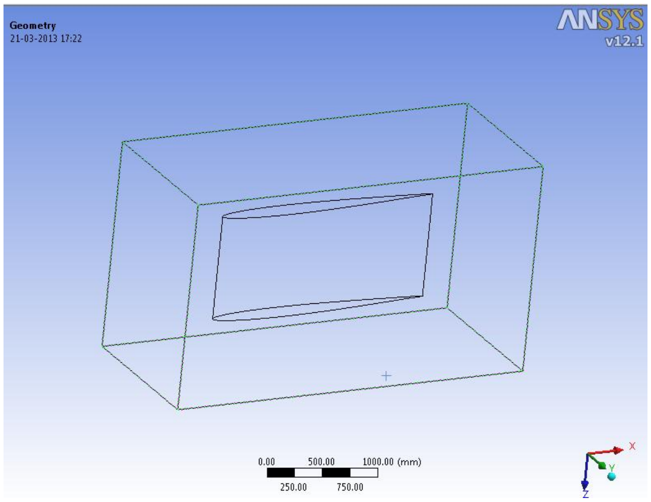


Fig. 2: Cavity of VAWT in ANSYS Workbench

Import IGES File in the ANSYS Workbench environment for meshing. Before meshing. We have to Verify Geometry for Clean-up.

C. Create Volume Mesh.

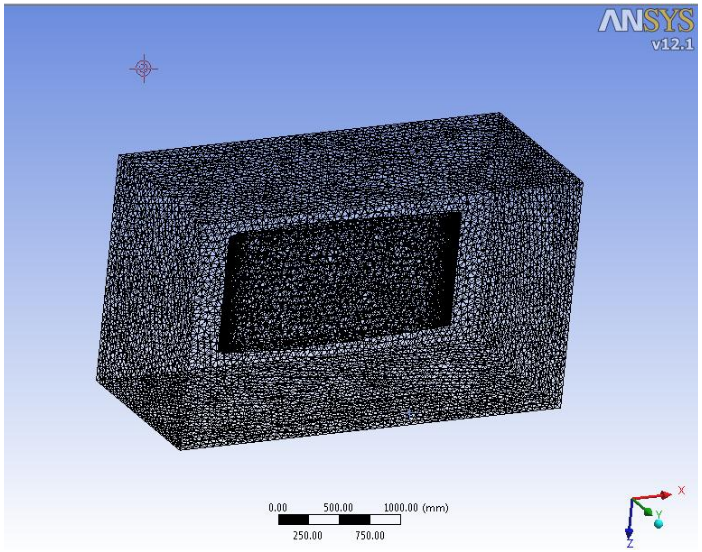
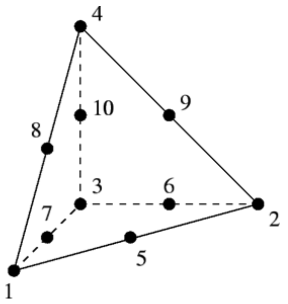


Fig. 3: Meshed Model of VAWT

After Verifying Geometry, We have to Create Volume mesh.



Elements Statics for this Meshing is as under:

Type of Element:- 10 node Tetrahedral
Total Number of Nodes = 277225
Total Number of Elements = 1566437

Mesh Preferences is CFX Mesh
Save file as VAWT. cmdb.

D. Import Mesh File into ANSYS CFX Pre

In general, a finite-element solution may be broken into the following three stages.

(1) Pre-processing: defining the problem
The major steps in pre-processing are (i) define key points/lines/areas/volumes, (ii) define element type and material/geometric properties, and (iii) mesh lines/areas/ volumes as required.

(2) Solution: assigning loads, constraints, and solving Here, it is necessary to specify the loads (point or pressure), constraints (translational and rotational), and finally solve the resulting set of equations.

(3) Post processing: further processing and viewing of the results.

In this stage one may wish to see (i) lists of nodal displacements, (ii) element forces and moments, (iii) deflection plots, and (iv) Pressure contour diagrams or temperature maps.

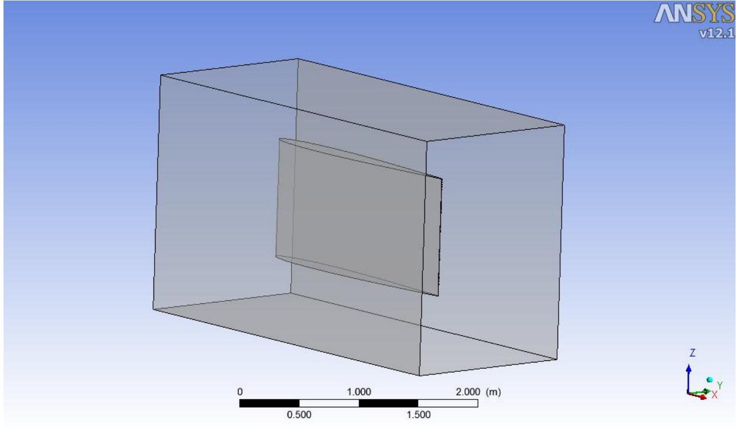


Fig. 4: Model in the ANSYS environment

After Creating Meshing file Import meshing file into CFX pre for pre-processing step.

E. Define Domain

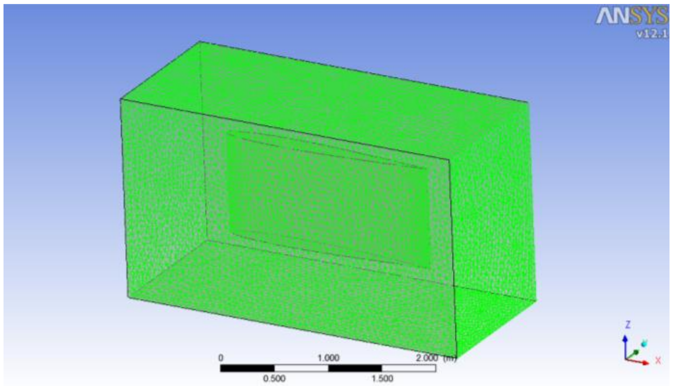


Fig. 5: Meshed model of the wind cavity

Domain defines type of fluid, Co-ordinate System, Reference Pressure, its Buoyancy Condition, Domain Motion, Heat Transfer Model, etc.

Here,
Domain Type: - Fluid Domain
Fluid: - Air Ideal Gas
Co-ordinate Frame: - Defaults
Reference Pressure: - 1 Atmosphere
Buoyancy Condition: - Non Buoyant
Domain Motion: - Stationary
Mesh Deformation: - None
Heat Transfer Model: - ISOTHERMAL
Turbulence: - K-epsilon

Where k is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. It has dimensions of (L2 T-2); for example, m2/s2.

ε is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate), as well as dimensions of k per unit time (L2 T-3) (e.g., m2/s3).

The k-ε model introduces two new variables into the system of equations. The continuity Equation is then:

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho U) = 0$$

and the momentum equation becomes

$$\frac{\partial \rho U}{\partial t} + \nabla \bullet (\rho U \otimes U) - \nabla \bullet (\mu eff \nabla U) = \nabla P' + \nabla \bullet (\mu eff\ \nabla U) T + B$$

Wall Function: - Automatic

F. Define Inlet and Outlet Condition as Below

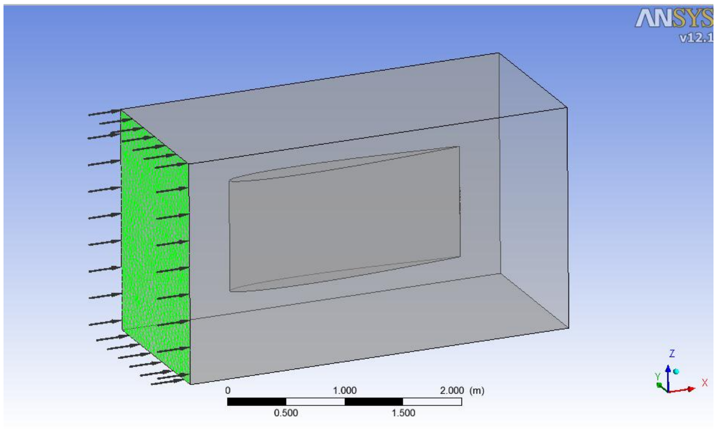


Fig. 6: Input conditions applied on the wind cavity

Inlet Condition is as under:
Fluid Regime: - Subsonic
Mass and Momentum:-
Normal Speed: - 8 m/s
Turbulence: - Medium

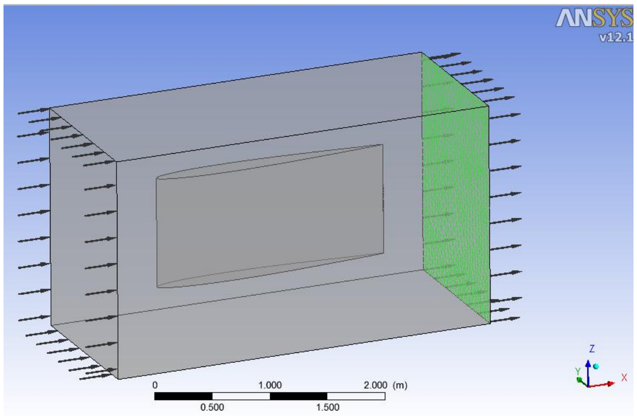


Fig. 7: Input and Output condition of the wind cavity

Outlet Condition is as under:-
Fluid Regime: - Subsonic

G. Go on Solver Option

Set Iteration to 100

H. Run Analysis

I. ANSYS CFX will Create Supersonic Nozzle.res file

J. Open CFX Post

K. Load .res file

VI. EXPERIMETAL ANALYSIS OF AIROFOIL

A. Experimental Investigation Method:

It has already been proved that the wind tunnel is the ideal location for systematic development work. However, tunnel dimensions for Wind Turbine Blades are problematic. Precise aerodynamic measurements can just about be made on full-size small blades in the largest wind tunnels available today. Larger blades create too larger an obstructions in the test section, both lengthwise and crosswise .they must therefore be studied in reduced scale.

The prerequisite for this is similarity in geometry and flow dynamics that is precise and detailed replicas of the originals and identical Reynolds numbers for models and full-size version.

Study of the flow over blade geometries can be performed by experiments in wind tunnels, as well as by numerical simulation. Wind tunnels have been extensively used for years in wind turbine aerodynamics in order to improve modern wind turbine design. In the past, experiments mainly involved the measurement of the aerodynamic coefficients and flow visualization over wind turbine blade. Performance on wind-tunnel experiments to measure the wakes of wind turbine blade are already been done. Also the experiment on performance of wind-tunnel tests using 1/5-scale basic wind turbine structures.

There are various types of Wind Tunnels: -

(1) Blower tunnel
(2) Suction tunnel
(3) Open jet tunnel
(4) Induction tunnel
(5) Supersonic tunnel

But for this work, slow speed suction wind tunnel is used to check the various effects of control parameters on the response parameters relative to the various aerodynamic bodies taken into this experimentation. Air is sucked in the tunnel through the contraction, honeycomb, test section, etc. by an exhaust fan. The air jet in the entire tunnel is slightly below atmospheric pressure; therefore for a given velocity in the test section the power required to drive the fan is comparatively less. Below figure shows the Slow speed Suction type Wind Tunnel.

Fig. 5.1: Wind tunnel

The specifications of various components are as follows: -

(1) Blower: The specifications of blower are as follows:

- *kW/HP-4/5*
- *Speed-3000*
- *Volts-220*
- *Amp-20*

(2) Nozzle:

The contraction or the nozzle feeds the test section with a jet of uniform velocity. The modal to be tested if fixed here with suitable supports. A transparent window of a strong glass is often provided on one or both the side walls on the test section. This facilitates in handling the modal and the instrument and also permits the optical measurement in the flow over the model surfaces.

The function of nozzle is as below:

- Accelerates the flow
- Makes the velocity distribution over the cross section of the flow more uniform.
- Reduces the intensity of the turbulence in the air stream.
- Serves to measure the wind speed in the test section.

(3) Test section:

It is the space where the model is to be kept and it is tested aerodynamically. The model to be tested if fixed here with suitable supports. A transparent window of a strong glass is often provided on one or both the side walls on the test section. This facilitates in handling the model and the instrument and also permits the optical measurement in the flow over the model surfaces.

(4) Power unit: This is the unit which consist of voltage control devices which supply the proper amount of voltage to the blower motor.

(5) Diffuser: The diffuser collects the flow from the test section and raises the pressure on the air for discharging it into the atmosphere or the return circuit in case of a close circuit tunnel. Boundary layer thickening and separation on account of the strong pressure gradients in the diffuser should be minimized. The diffuser throat is often made flexible, this allows the throat to be varied for starting and running conditions. After starting the diffuser throat area is reduced for optimum running conditions.

(6) Pressure measuring unit: This unit measures the pressure differences of the model. Through this unit the drag coefficient can be calculated by using pressure difference.

(7) Honey comb: The flow from the compressor/blower or fan is settled in a large chamber called the settling chamber, this is provided with wire gauges and arrays of honeycombs to straighten the flow and remove irregularities in it.

Experiment

Air inlet Angle	*Inlet Velocity m/s*	*Exit Velocity m/s*
20.1	*8*	*18.04*

VII. CONCLUTION

CFD is capable of designing the VAWT with higher degree of accuracy. It can also be used for the optimization of blade design. Moreover, flow field around various configurations' blades can also be visualized with the help of CFD. It has not only accelerated the design process of VAWT but also has brought down the overall cost of designing.

Various vertical axis wind turbines can offer solution to the energy requirements ranging from 2 kW to 4 MW with a reasonable payback period. Coefficient of power can be maximized by selecting a suitable TSR range for various configurations. The parameter like air strike angle and aerofoil of the blade, wind speed, tip speed ratio, chord length are one of most effecting parameter for the vertical axis wind turbine. By the analysis using this parameter can become useful in the performance optimization of the vertical axis wind turbine.

REFERENCE

[1] Marco Raciti Castelli, Alessandro Englaro, Ernesto Benini, "The Darrieus wind turbine: Proposal for a new performance prediction model based on CFD", Science Direct, 20 May 2011.
[2] Francesco Balduzzi, Alessandro Bianchini, Ennio Antonio Carnevale, Lorenzo Ferrari, Sandro Magnani "Feasibility analysis of darrieus vertical-axis wind turbine installation" Science Direct, 8 December 2011.
[3] M. EI-Samanoudy , A.A.E. Ghorab, Sh.Z. Youssef, "effect of some Design parameters on the performance of giromill vertical axis wind turbine" Science Direct, 5 August 2010.
[4] K. Pope, V. Rodrigues, R.Doyle, A. Tsopelas, R. Gravelsins, G.F. Naterer, E.Tsang, "effect of Stator vanes on power coefficient of a zephyr vertical axis wind turbine",Science Direct, 11 October 2009.
[5] Robert Howell, Ning Qin, Jonathan Edwards, Naveed Durrani, "wind tunnel Numerical study of a small vertical axis and wind turbine", Science Direct, 20 July 2009.