International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 08
July 2019

Design and computational flow field analysis of horizontal axis wind turbine

Ansari Umair Ahmed (M.tech) [1] , Mr. N. Nagunaik [2], Mr. G. Satish Kumar [3]

Department of Mechanical Engineering, Narsimhareddy Engineering College, Hyderabad, Telangana-500100, India

umairansari111@gmail.com[1], nagunaikrhl@gmail.com[2]

ABSTRACT

The proposed objective of the CFD analysis is to investigate the flow field around a horizontal axis wind turbine rotor and calculate the turbine's power. In this study, a full three dimensional computational fluid dynamics method based on Reynolds Averaged Navier Stokes approach was used. The wind turbine with rotor diameter of six meters has three blades. In a moving reference frame system, one third of the wind turbine rotor was modeled by means of 120o periodicity. The coefficient of power curve obtained from the CFD results is compared with experimental data obtained by NREL Phase VI rotor experiment. The numerical result for the coefficient of power curve shows close agreement with the experimental data. The simulation results include the pressure distribution, velocity distribution along the flow direction, turbulent wake behind the wind turbine, and the turbine's power. The discussion includes the effect of wind speed on turbine's power.

1. INTRODUCTION

a. World Wind Energy

The proposed purpose of the CFD evaluation is to examine the float discipline round a horizontal axis wind turbine rotor and calculate the turbine's vigor. On this learn, a full three dimensional computational fluid dynamics process centered on Reynolds Averaged Navier Stokes method was once used. The wind turbine with rotor diameter of six meters has three blades. In a moving reference body process, one 1/3 of the wind turbine of power curve bought from the CFD end result is when compared with experimental expertise obtained with the support of NREL section VI rotor scan. The numerical outcomes for the coefficient of vigour curve suggests shut agreement with the experimental know-how. The simulation outcome incorporate the stress distribution alongside the flow path, turbulent wake at the back of the wind turbine, and the turbine's power. The dialogue involves the effects of wind pace on turbine's vigour.

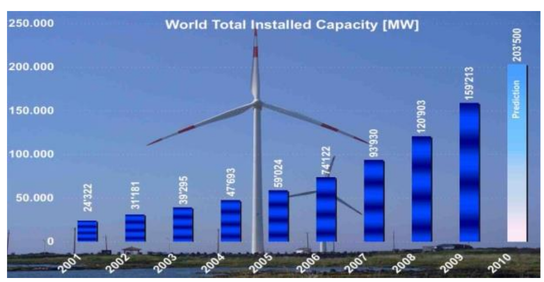


Figure 1. World total Installed Capacity by the end of 2009 (Data source: WWEA, www.wwea.org)

b. Motivation

Wind vigor represents a low density source of power. Maximizing the effectivity of converting wind energy into mechanical form of power has greatest importance of making wind vigour economically possible. The talents and working out of rotor aerodynamics, the design of blades shapes improves the overall performance of the latest mills. Wind turbine technological know-how is based on drive distributions on the blades of turbine's rotor, leading to mechanical torque at the shaft. The shaft then transfers the torque from the blades to the generator. In latest wind generators aerodynamic driving drive is traditionally the raise force rather than drag drive like because it was in historical crusing ships. Study work conducted in this field has brought a large improvement within the total efficiency in energy conversion method of wind turbines. The potential to foretell the downstream wakes of the go with the flow discipline is a giant aspect for deciding upon the interactions between mills blade.

There are three strategies to be had to research the glide round and downstream of wind turbines [3]:

1. Discipline checking out is the person who supplies correct outcome but is extremely tricky and high priced.

2. Analytical and semi-empirical models, which undertake the simplified assumptions and are consequently not universally riskless.

3. Computational Fluid Dynamics (CFD), is the satisfactory substitute to direct testing. The aim of this thesis is to gain knowledge of the aerodynamics of Horizontal Axis Wind Turbine through numerically solving governing equations using finite-quantity system and Reynolds Averaged Navier-Stokes process.

2. LITERATURE REVIEW

Wind generators efficiency and its rotor characteristics was the field of investigation for a long time. In 1915, Lanchester [21] was once the primary to predict the maximum efficiency of an superb wind turbine of 59.3%.

In 1920, German scientist Betz and Russian scientist Joukowsky, derived this maximum efficiency independently Being unaware of Lanchester's findings. Nonetheless, the limit is known as Betz limit. The essential wreck-through in rotor predictive ways was once done by means of Glauert [25], in 1935 who formulated the Blade element Momentum (BEM) method . The approach used to be centered on momentum stability equations for man or woman annular movement tubes passing by way of the rotor. In BEM, the turbine blade is split into separate blade segments and analyzed from a two-dimensional standpoint. At present, the industrial rotor design codes are nonetheless headquartered on BEM [3, 4 and 5]. Aerodynamic modeling of HAWT rotors by way of a conventional engineering ways has reached the point the place no extra improvement will also be expected and not using a full working out of the float physics [6].

Extensive use of the numerical experiences on all HAWT aerodynamics features, carried out on many exceptional stages, ranging from BEM approaches built-in with the aid of CFD calculations to full 3D Navier-Stokes models grew to become really well known manner to predict the efficiency and characteristics of brand new wind turbines. Many authors have used the generalized Actuator Disk process that represents roughly an extension of the BEM procedure, built-in in an Euler or Navier-Stokes frame [7, 8]. The procedure describes forces which can be disbursed evenly alongside the azimuthal direction; The 3D Navier-Stokes solver has been combined with the so-referred to as Actuator Line process, in which the loading is dispensed along strains representing the blade forces [4, 5, and 8].

Within the few earlier years, Sankar and co-staff [9, 10, and 11] developed a hybrid Navier-Stokes/Full-knowledge/Free Wake process, probably for predicting the 3D viscous waft over helicopter rotors. This system has recently been accelerated to maintain the HAWT waft fields. The computational domain is split in one of a kind regions, every one may also be solved with the aid of the right technique: Navier-Stokes resolution close the blades, skills waft representation on the outer discipline and a set of the vortex methods for modeling vorticity discipline. Full 3-dimensional computations employing Reynolds-averaged Navier-Stokes (RANS) equations have been carried out by Duque [12], Ekaterinaris [13], Sørensen and Michelsen [14].

Risø and Denmark Technical university carried out a couple of numerical investigations on HAWT making use of their Navier-Stokes solver

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 08
July 2019

EllipSys 2d/3D, coping with design of rotors, overall performances, blade sections [4, 15, and 16], severe operation stipulations [17] and the tip form [18].

In 2008, Mandas, et al. [19] at the institution of Cagliari in Italy used the commercial code Fluent to participate in the unique analysis of Horizontal Axis Wind Turbine glide. The constant float area around the remoted rotor of a middle-sized HAWT is expected in a non-inertial reference frame, using each the Spalart- Allmaras [27] and the Menter's okay-ω SST [20] turbulence specifying a consistent axial wind pace on the inlet. In a similar way, in this thesis, the 3D conduct of the upstream, downstream float and the wake, will be investigated by using finite – quantity method and Reynolds Averaged Navier-Stokes strategy.

3. AERODYNAMICS OF THE WIND TURBINE

a. Basic Definitions Of Wind Turbines

The following parameters were used in calculations of turbine's power coefficient:

Tip Speed Ratio

$$\lambda = \frac{\Omega r}{V_\infty} \qquad [2.1.1]$$

Tip speed ratio (also called TSR) is defined as the ratio between the tip speed (tangential velocity) and the undisturbed wind speed entering the turbine.

It is a dimensionless variety and represents a very important turbine parameter. Typical values of the tip pace ratio for the cutting-edge turbines are 6 to 8.

Induction Factor

$$a = \frac{V_0 - V}{V_0} \qquad [2.1.2]$$

The fractional decrease in the wind velocity between the free stream and the rotor plane can be expressed in terms of an axial induction factor, a:

Where, V is the velocity at the disk (see Appendix A) and it is defined by

$$V = \frac{1}{2} * (V0 + V3) \qquad [2.1.3]$$

V0 and V3 are free stream and the downstream velocities respectively. The amount of axial induction factor determines the amount of power extracted by the turbine.

Power Coefficient

$$C_p = \frac{P}{\frac{1}{2}\rho A V^3} = \frac{\textit{Power Extracted by Rotor}}{\textit{Power Available in the Wind}}$$

Finding the greatest practical cross-sectional area of the wind creates the opportunity to extract the greatest amount of energy. The practical efficiency of a wind turbine is usually represented by the power coefficient Cp, defined as the power extracted by rotor to power available in the wind

[2.1.4]

The power coefficient CP, is defined as extracted power over the total available power can be similarly defined in terms of axial induction factor a as:

$$C_p = 4a\,(1 - a)^2 \qquad [2.1.5]$$

Lanchester–Betz–Joukowsky limit [21] shows that the actual turbine can't extract greater than fifty nine.3% of the vigor in an undisturbed tube of air of identical area. In apply, the fraction of vigour extracted will invariably be less considering that of mechanical imperfections. A great portion is 35-40% of the energy in the wind below excellent conditions, in spite of the truth that divisions as high as half of have been asserted [19]. A turbine which separates forty percentage of the power in the wind is removing round 66% of the sum that will be extricated via a excellent turbine

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 08
July 2019

Highest Cp is decreased by using:

1. Wake rotation in the back of (downstream) of the rotor.

2. A non-uniform pressure distribution within the aircraft of turbine

3. Tip losses and quantity of blades

4. Aerodynamic drag

Wake rotation has a excessive affect on the effectively of the turbine and it's much related to this

$$P = \frac{1}{2}\rho A V^3 \quad [2.1.7]$$

thesis. An overview of this phenomenon will be offered in the section 2.2. Generators energy Wind vigor is proportional to the cube of the wind's pace. This relationship is provided mathematically with the aid of the next equation:

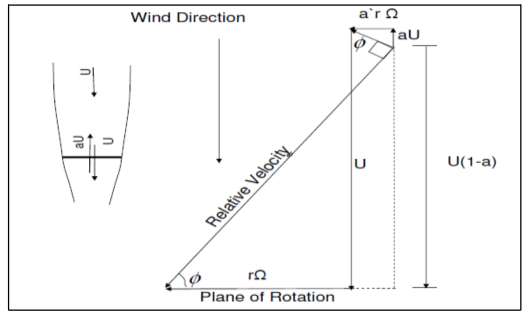
b. Rotating Wake Effect

The impact of the rotating wake may also be estimated by extending the Betz analysis to a 2-D model in the radial course. The float some distance upstream is only axial; nevertheless, there's discontinuous leap in angular pace throughout the rotor airplane for the reason that torque is exerted on the rotor. Even as the flow imparts a torque to the disk, the disk in turn imparts an equal and opposite torque to the flow. Thus, if the disk rotates with angular speed Ω, the go with the flow rotates within the opposite course, say with angular speed ω. Now, keep in mind an observer relocating with the disk with angular velocity Ω. The observer sees that the flow at the back of the rotor is moving with angular pace Ω - (-ω) or Ω + ω within the reverse direction. Two extra expressions are more often than not introduced: an angular induction factor, a', and neighborhood tip speed ratio λr, defined as:

$$a' = \frac{\omega}{2\Omega} \quad [2.2.1]$$

$$\lambda_r = \frac{\Omega r}{V_0} = \lambda \frac{r}{R} \quad [2.2.2]$$

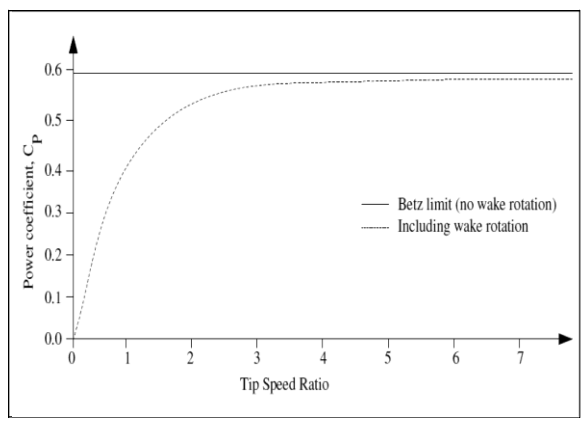
Glauert [25] derived the expression for the pressure, p2-p3, by applying Bernoulli's energy equation [26]



with respect to observer, before and after the disk.

Figure 3.1. Flow velocity diagram at an annulus in HAWT rotor disk

Figure 3.2 depicts the theoretical maximum power coefficient as a function of tip speed ratio with and



without wake rotation effect [8].

$$C_{p_{max}} = C_{p_{Betz}} = \frac{16}{27} = 0.593 \quad [2.1.6]$$

Figure 3.2 Power coefficient curve with and without wake rotation

Typical turbine performance from real devices is presented in the Figure 4 bellow which depicts the power coefficient plotted against tip speed ratio, λ [22].

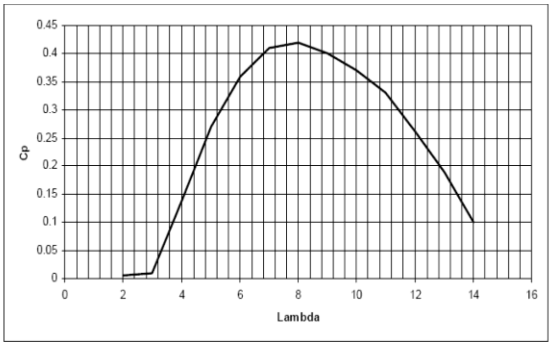


Figure 3.3 Cp vs. λ curve for actual wind turbine

c. Airfoil Characteristics

A quantity of phrases are used to characterize an airfoil. The imply camber line is the locus of a factor halfway between higher and cut back surfaces of the airfoil. The most forward and rearward ends of this locus line are the main facet and the trailing facet respectively. The road connecting the main side and the trailing side is the chord line and the distance between the two edges is referred to as chord, c. Camber is the distance from the imply camber line to the chord line, measured perpendicular to the chord line. The thickness t, is the distance between the higher and slash surfaces, also measured perpendicular to the chord line. The angle of attack, α, is the attitude between the relative water speed and the chord line. A graphical representation of an airfoil part is proven in determine Fig 5.

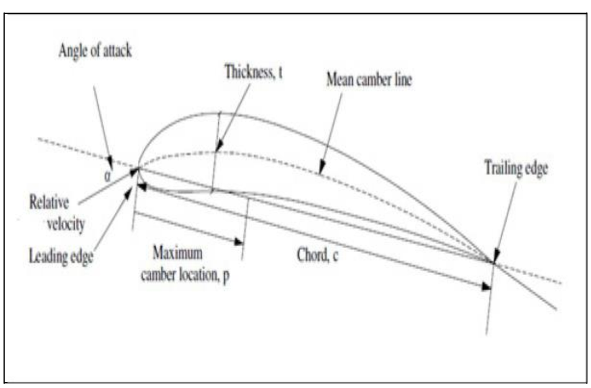


Figure 3.4.Airfoil section

d. Forces Acting On An Airfoil Section

When an impending flow encounters an airfoil part, the forces act on it. The forces can be resolved into average force, FN (as shown in determine 6), along the float path and thrust force toes, orthogonal to it. On the other hand they can also be resolved into elevate drive FL, average to the relative pace of the glide and drag forces FD, alongside the glide.

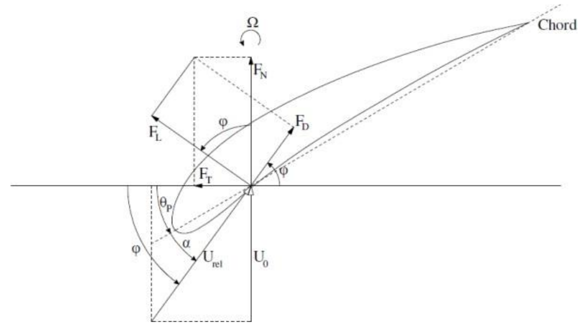


Figure 3.5 Forces acting on airfoil

4. COMPUTATIONAL FLUID DYNAMICS (CFD)

a. CFD Principles and Advantages

Computational Fluid Dynamics may also be explained as numerical technique to the differential governing equations of fluid flows with the help of computer systems. CFD solvers are based on way to any of the following three principal ideas which might be :

• Conservation of mass

• Newton's 2nd legislation (conservation of momentum)

• Conservation of vigour

These principles can also be expressed in terms of the mathematical equations, customarily in imperative and partial differential equation forms. CFD is a instrument to exchange these equations with the discretized algebraic equations, which in flip are solved in forms of numbers; thus the top made of simulation is a collection of numbers [30]. Computational fluid dynamics outcome are analogous to the wind tunnel outcome got in laboratories: they both provide the set of knowledge for a given float configurations at exceptional conditions like laminar or turbulent, constant or unsteady. Nonetheless, in contrast to wind tunnel, associated with high price and maintenance, CFD results are more affordable. A number of valuable benefits are achieved with the aid of following the CFD process utilized to fluid dynamic problems: CFD is faster and more cost effective. Huge reduction of time and cost for fixing the one of a kind issues compared to the average procedures.

• Wind tunnels are limited in size, thus full size analysis are tough to perform for a enormous techniques (e.g., world's greatest wind turbine ENERCON E-126 has 126 m diameter rotor and it is set 200 m high). A CFD study is a favorable alternative in this case.

• CFD supplies in a unique solution, enabling robust evaluation of the mannequin at every vicinity at any time instantaneous.

• With the brand new advancements within the technologies, turbulent items and solution schemes, numerical units of the bodily issues have good accuracy and reliability.

• Usually (rather then mills) the prediction of the fluid flows does no longer require powerful workstations and oftentimes private desktops maybe adequate.

b. CFD Code

4.b.1. Preprocessor

In preprocessor segment the bodily problem is changed into a mathematical mannequin. The computational domain is outlined and subdivided into smaller parts referred to as grid or mesh. Fluid is outlined and boundary stipulations are set. Due to the fact that the CFD answer depends in the neighborhood on number of factors or grid, meshing of the area and geometry could be very principal for reaching accuracy in outcome. Quite simply, higher the number of grids greater is the accuracy of the evaluation. Mesh spacing is very so much accurate to more advantageous by way of striking finer grids close the area of excessive variable gradients and coarsen the place outcome of the flow is no longer much primary. Moreover, satisfactory of mesh and variety of mesh affects the result in a higher scale and accordingly a detailed attention has to be paid to the mesh parameters like skewness or aspect ratio. Result of mesh excellent and style is defined later in chapter. In preprocessor stage geometry is created, masses and boundary stipulations are then set.

4.b.2. Solver

The numerical answer algorithm is the core of CFD code. CFD solvers work with the following procedure:

• Modeling the drawback unknowns

• Discretising the governing equations for the fluid flows

• Solving the algebraic process of equations.

4.b.3 Postprocessor

Postprocessor includes the final analysis of the solution results. The solver outputs the set resolution variables in varieties of graphs and contours. Area and grid visualizations, vectorial plots, linear, surface and volume integrals, tracking route- strains, dynamic representations, and animations are all constituents of the put up-processing part.

c. ANSYS Fluent

CFD Finite Volume Approach

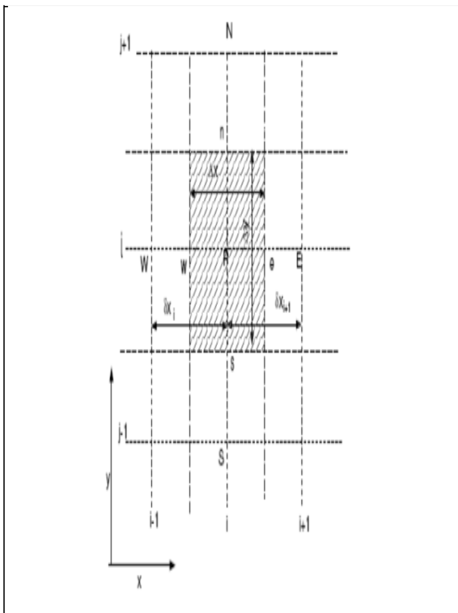
The commercial code Fluent solves the governing equations for the conservation of mass and momentum, and (when proper) for power and different scalars, akin to turbulence and chemical species. In each circumstances the control-quantity-headquartered process is used [45, 46]. Discretization steps are as follows:

• Division of the domain into discrete control volumes utilizing computational grid.

• Integration of the governing equations on the individual control volumes to assemble algebraic equations for the discrete elegant variables (unknowns), comparable to velocities, strain, temperature, and conserved scalars.

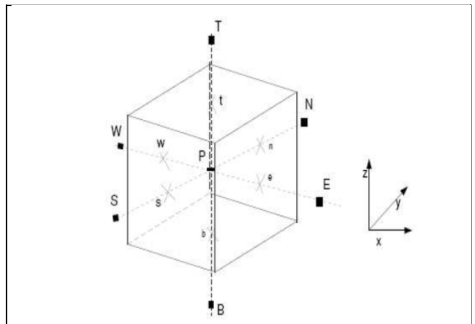
• Linearization of the discretized equations and the resolution of the resultant linear equation system, to yield updated values of the dependent variables.

Figure 7 under, indicates the small volume factors around the node in second and 3D Cartesian coordinates. It makes use of a co-located grid, that means that all the go with the flow parameters are saved in the telephone-facilities. Tactics will also be without problems consequently parallelized on more than one computer nodes.



2-D Cartesian grid with general notation in finite volume method

(a)



Control volume in 3-D with Cartesian coordinates

(b)

Figure 4.1 Finite-volume representations in Cartesian coordinates: a) 2D b) 3D

Finite-volume procedure is a conservative: the flux going out by means of a face of one control volume is the same as the flux coming out into the adjacent manipulate quantity trough the same face.

d. Numerical Solvers

Fluent is a business 2nd/3D mesh solver, which adopts the multigrid solution algorithms. Two numerical solver applied sciences are available in Fluent:

• Pressure-centered solver

• Density-established solver The first solver was once developed for the low-velocity incompressible flows, whereas the 2d used to be created for the high-velocity compressible flows resolution. In the reward be taught, which includes the incompressible flows, the strain-situated procedure was therefore desired. Each methods are actually applicable to a large range of the flows (from incompressible to enormously compressible), but the origins of the density-situated formulation can provide it an accuracy (i.E. Shock decision) competencies over to the stress-centered solver for top-velocity compressible flows.

4.d.1 Pressure Based Solver

The stress-based solver employs an algorithm which belongs to a common category of ways referred to as the projection procedure. Within

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 08
July 2019

the projection procedure, the constraint of mass conservation (continuity) of the pace subject is accomplished through solving a strain (or stress correction) equation. The strain equation is derived from the continuity and the momentum equations in this kind of means that the pace discipline, corrected by way of the pressure, satisfies the continuity. When you consider that the governing equations are nonlinear and coupled to one an extra, the answer approach for that reason entails iterations where the complete set of governing equations is solved many times except the resolution converges. [31] Fluent supplies three one-of-a-kind solver formulations:

• Segregated

• Coupled implicit

• Coupled specific the way where the governing equations are linearized may just take an "implicit" or "express" form with recognize to the dependent variable (or set of variables) ofthe curiosity.

4.d.2. Density-Based Solver

The density-founded solver accordingly solves the governing equations of continuity, momentum, and (the place proper) vigour and species transport simultaneously (i.E., coupled collectively). Governing equations for extra scalars will then be solved later on and sequentially (i.E., segregated from one another and from the coupled set). In density-established resolution approach, you'll be able to remedy the coupled process of equations (continuity, momentum, power and species equations if on hand) using, either coupled-express components or the coupled-implicit formulation. If you happen to decide upon the implicit alternative of the density-based solver, each and every equation within the coupled set of governing equations is linearized implicitly with recognize to all of the dependent variables in the set. Within the specific choice of the density-headquartered solver, each equation in the coupled set of the governing equations is linearized explicitly.

5. MESH GENERATION

5.1 Structured Grid Methods

Right here mesh grid is laid out in a typical repeating pattern called a block. These varieties of grids utilizes tquadrilateral factors in 2nd and hexahedral elements in 3D in a computationally rectangular array. Relatively excellent structured grid turbines make use of subtle elliptic equations to automatically optimize the form of the mesh for orthogonality and uniformity.

5.2 Hybrid Grid Methods

Hybrid grid ways are designed to take skills of the constructive elements of both volumes structured and unstructured grids. Hybrid grids can include hexahedral, tetrahedral prismatic and pyramid elements in 3D and triangles and quadrilaterals in 2d. Hexahedral elements are quality close stable boundaries (the place waft field gradients are high) and afford the customers a excessive degree of manipulate. Prismatic factors (most of the time triangles extruded into wedges) are valuable near

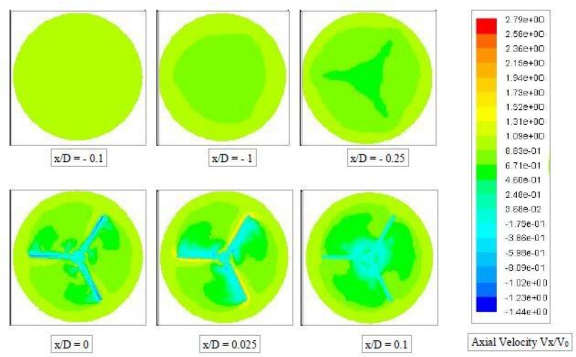


Figure 27. (a) Axial velocity contours at different x/D locations for TSR=7 with stream velocity V_0=10 m/s

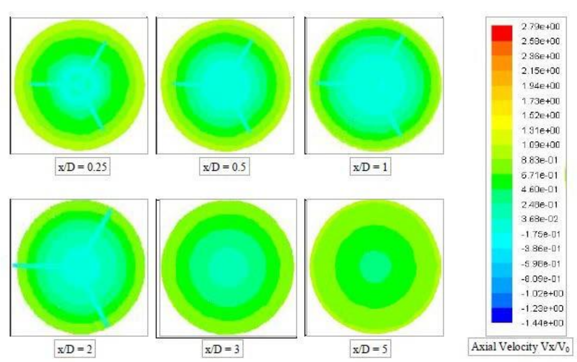


Figure 27. (b) Axial velocity contours at different x/D locations for TSR=7 with stream velocity V_0=10 m/s

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 08
July 2019

wall gradients. In most all instances, tetrahedral elements are used to fill the rest.

5.3 Mesh Quality

The high-quality of the mesh performs a n principal position in the accuracy and steadiness of the numerical computation. The attributes associated with the mesh quality are node factor distribution, smoothness, and skewness.

6. RESULTS AND DISCUSSIONS

7.1 Flow Visualization

This chapter offers with the flows before and after the blade, it's surrounding subject and the wake of the turbine. A assessment of axial velocity distribution for various TSR`s is presented. A case TSR 7 is taken for finding out the waft traits. The inlet stream speed is constant at 10 m/s and the blade has a rotational speed of ω = -23.33 rad/s. The moving reference frame was once modeled with a fluid rotating around the blade. Turbine itself offered as moving wall with rotational speed of zero rad/s with appreciate to rotating fluid (air on this case). The axial, radial and tangential pace contours are plotted in exceptional sections along the x-axis (flow course).

Figure 6.1 shows the wake expansion along the flow at y = 0 inclined plane, which cuts through the turbine blade at angle of 60o to the z-axis

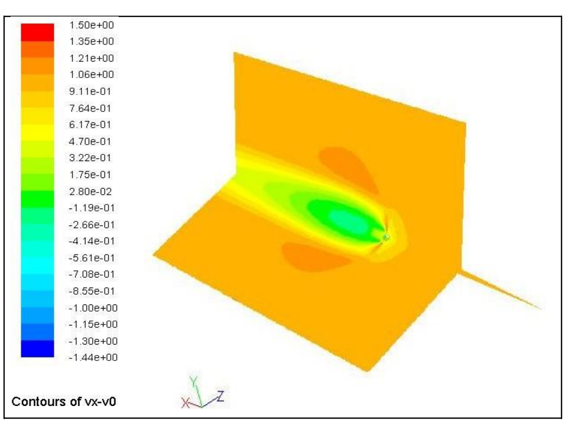


Figure 7.2. Axial velocity distributions along the flow direction As go with the flow moves downstream, it may be noticeable that wake expands alongside the course of the flow.

Nonetheless, magnitude of speed decreases inside the move tube. Green area right after the turbine suggests very low and even poor velocities precipitated by means of the hub bluntness and mixing flows. Mild yellow field corresponds to speed magnitude of about 6.17 m/s on the outflow, which saved at distance of 5D after the rotor.

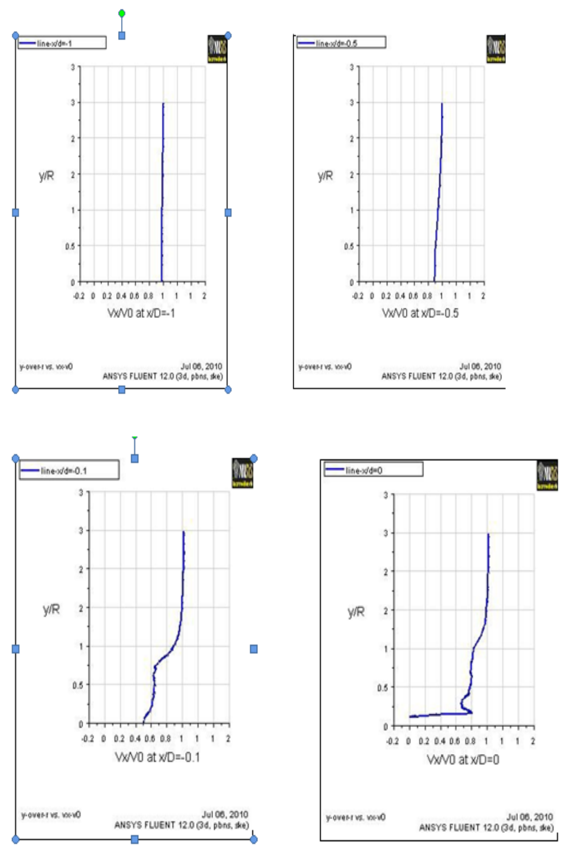


Figure 7.3. Axial velocity plots along the radial lines at various x/D locations: V=10 m/s, TSR=7

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 08
July 2019

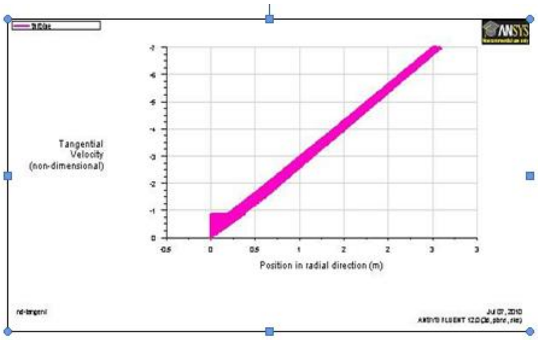


Figure 7.4. Tangential velocities along the blade

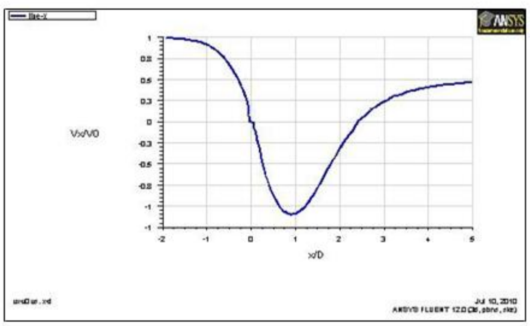


Figure 7.4. Axial velocities along the x-direction

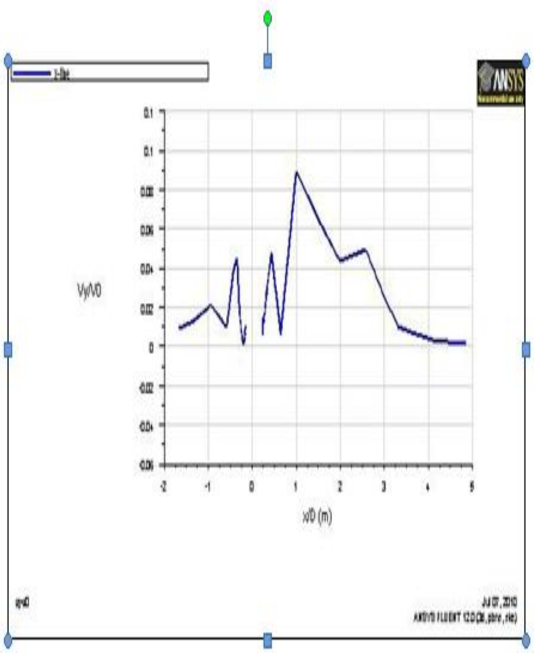


Figure 7.5. Radial velocities along the x- direction

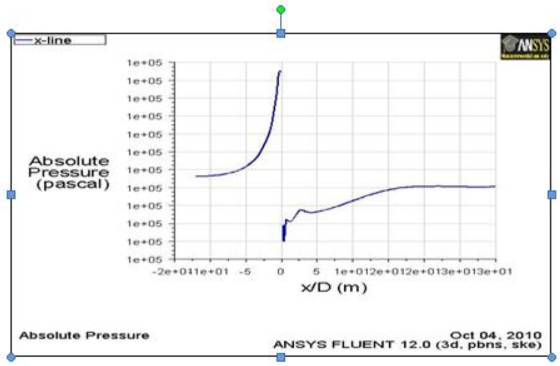


Figure 7.6 Static pressure distributions along the flow direction

CONCLUSION

In this research work, a methodic study of a flow field around horizontal axis wind turbine was investigated using RANS method. The flow model has been validated against the results of a Phase VI rotor experiment conducted by NREL. The turbine power coefficient obtained in this study using CFD analysis was in a good agreement with experimental data. Near and far wake studies have been carried out. The variations of the velocity components, axial, radial and tangential have been presented and investigated. The reduction in the axial flow velocity profiles as the flow moves along is in agreement with actuator disk theory. Axial, radial and tangential velocity contours at different x/D locations are also presented. It was found out that tangential velocity effects nullify faster than the radial velocity effects. Parametric study was also performed to investigate the turbines power at different free stream wind velocities value. Turbines power increases as wind speed increases; however, at low speeds the predictions are underestimated. The turbulent viscosity, turbulent intensity and pressure distribution plots were also presented. The future work is recommended to:

• Investigate the pressure distribution on the blade surface by performing 2D analysis of the airfoil section.

• Predict the lift and drag coefficients at different angles of attack.

REFERENCES

[1] World Wind vigor association (WWEA), World Wind power report 2009 http://www.Wwindea.Org

[2] World energy Outlook 2009 http://www.Iea.Org

[3] Vermeer LJ, Sørensen JN, Crespo A. Wind Turbine Wake Aerodynamics; growth in Aerospace Science, 2003; Vol. 39; 467-510.

[4] Sørensen JN, Shen WZ. Numerical Modeling of Wind turbine Wakes; J. Fluid Engineering 2002; Vol.124; 393-399.

[5] Ivanell SSA. Numerical computations of wind turbine wakes; Technical reviews from KTH Mechanics, Royal Institute of technology; Stockholm, Sweden; 2005.

[6] Kang S, Hirsch C. Features of the 3D go with the flow around wind turbine blades headquartered on numerical options; complaints from EWEC 2001, Copenhagen.

[7] Alinot C, Masson C. Aerodynamic simulation of wind generators running in atmospheric boundary layer with more than a few thermal stratifications; AIAA Paper 2002-0042.

[8] Mikkelsen R. Actuator disc methods utilized to wind turbines; Dissertation submitted to the Technical school of Denmark in partial success of the requisites for the measure of PhD in Mechanical Engineering; Lyngby, 2003.