Aerodynamic analysis of 1.5MW horizontal wind turbine blade

Nikita Patil[1],Aditya Murkute[2], V. H. Waghmare[3]

[12]*UG Student,Department of Mechanical Engineering, Smt. KashibaiNavale College of Engineering, Pune.*

[3]*Assistant professor,Department of Mechanical Engineering, Smt. KashibaiNavale College of Engineering, Pune.*

***Abstract*: The paper presents a detailed design andaerodynamic analysis of small power wind turbine blade, including airfoil selection, pitch angle of blade tip.The aerodynamic simulations were performed using a Computational Fluid Dynamics (CFD) method based on the steady-state 1-way FSI (Fluid-Structure Interaction) analysis.The commercially available software FLUENT is employed for calculation of flow field using Navier-Stokes equation in conjunction with the k-omega shear stress transport (SST).The obtained results are verified using numerically calculated data with analytical data.**

***Keywords*:wind turbine blade, aerodynamic, computational fluid dynamics, FLUENT, Navier stokes equation.**

I. INTRODUCTION

Wind energy is one of the renewable energy sources and wind turbines are used to generate electrical energy by using the kinetic energy of wind. Power has been extracted from the wind over hundreds of years with historic designs, known as windmills, constructed from wood, cloth and stone for the purpose of pumping water or grinding corn. Historic designs, typically large, heavy and inefficient, were replaced in the 19th century by fossil fuel engines and the implementation of a nationally distributed power network. A greater understanding of aerodynamics and advances in materials, particularly polymers, has led to the return of wind energy extraction in the latter half of the 20th century. Wind power devices are now used to produce electricity, and commonly termed wind turbines. The orientation of the shaft and rotational axis determines the first classification of the wind turbine. A turbine with a shaft mounted horizontally parallel to the ground is known as a horizontal axis wind turbine or (HAWT). A vertical axis wind turbine (VAWT) has its shaft normal to the ground as shown in figure 1.

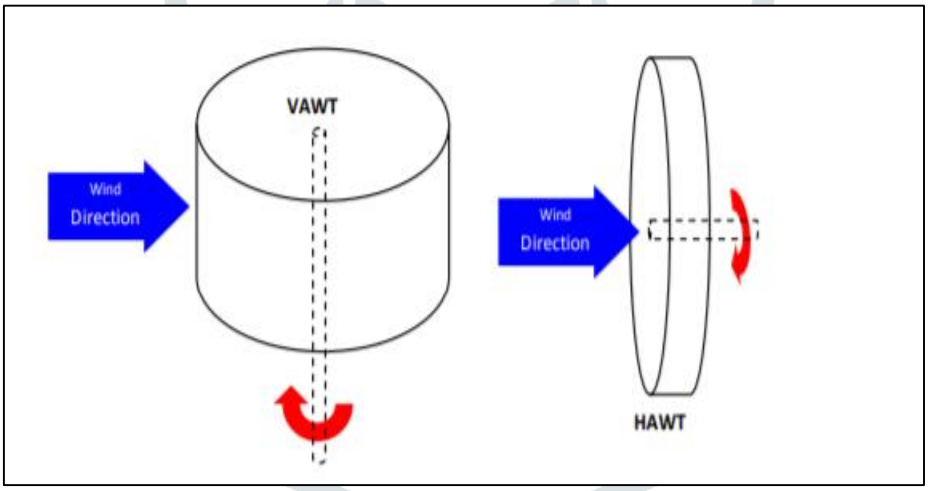

Fig .1 Alternative configurations for shaft and rotor orientation.

The two configurations have instantly distinguishable rotor designs, each with its own favorable characteristics *[1]*. The discontinued mainstream development of the VAWT can be attributed to a low tip speed ratio and difficulty in controlling rotor speed. Difficulties in the starting of vertical turbines have also hampered development, believed until recently to be incapable of self-starting *[2]*. However, the VAWT requires no additional mechanism to face the wind and heavy generator equipment can be mounted on the ground, thus reducing tower loads. Therefore, the VAWT is not completely disregarded for future development. A novel V-shaped VAWT rotor design is currently under investigation which exploits these favorable attributes *[3]*. This design is currently unproven on a megawatt scale, requiring several years of development before it can be considered competitive. In addition to the problems associated with alternative designs, the popularity of the HAWT can be attributed to increased rotor control through pitch and yaw control. The HAWT has therefore emerged as the dominant design configuration, capitalized by all of today's leading large-scale turbine manufacturers.Wind turbine design is the process of defining the form and specifications of a wind turbine to extract energy from wind. A wind turbine installation consists of the necessary system needed to capture the wind's energy. For reasons of efficiency, control, noise and aesthetics the modern wind turbine market is dominated by the horizontally mounted three blade design, with the use of yaw and pitch, for its ability to survive and operate under varying wind conditions.

II. BLADE GEOMETRY

The design the geometry of wind turbine the diameter of wind turbine rotor is chosen considering the aerodynamic performance, strength and stiffness condition, power generated.The blade is 43.2 meters long and starts with a cylindrical shape at the root and then transitions to the thick airfoils S818, S825 and S826 for the root, body and tip, respectively. This blade also has pitch to vary as a function of radius, giving it a twist and the pitch angle at the blade tip is 4 degrees.The turbulent wind flow towards the negative z-direction at 12 m/s which is a typical rated wind speed for a turbine this size. This incoming flow is assumed to make the blade rotate at an angular velocity of-2.22 rad/s about the z-axis (the blade is thus spinning clockwise when looking at it from the front, like most real wind turbines). The tip speed ratio (the ratio of the blade tip velocity to the incoming wind velocity) is therefore equal to 8 which is a reasonable value for a large wind turbine. Figure 2 shows the 3D model of blade designed in ANSYS 16.0.

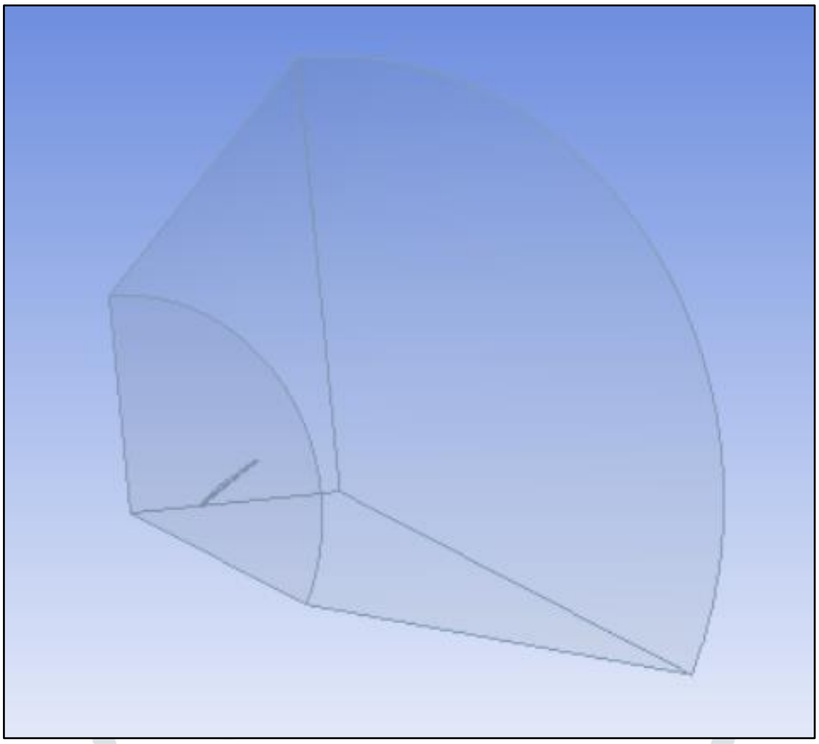

Fig.2 Blade geometry in ANSYS16.0

III. MATHEMATICAL MODEL

The governing equations are the continuity and Navier-Stokes equations. These equations are written in a frame of reference rotating with the blade. The equations are as follows:

Conservation of mass:

$$\frac{\partial p}{\partial t} + \nabla \rho . Vr = 0$$

Conservation of momentum (Naviers stokes equation):

$$\nabla . (\rho Vr Vr) + \rho(2\omega \times Vr + \omega \times \omega \times r) = -\nabla p + \nabla \tau r$$

Where Vr is the relative velocity when viewed from moving frame and ω is the angular velocity.

IV. CALCULATIONS

The velocity, v, on the blade follow the formula,

$$v = r * \omega$$

Plugging in our angular velocity of -2.22 rad/s and using the blade length of 43.2 meters plus 1 meter to account for the distance from the root to the hub,

$$v = 98.12 m/s$$

The specification sheet of this turbine states the rated power of turbine to be 1.5 MW, the rated wind speed to be 11.5 m/s and the rotor diameter to be 82.5 m.

the rated power calculated as:

$$Cp = \frac{rated\ power}{wind\ power} = \frac{rated\ power}{0.5\rho A v^3} = \frac{1.5MW}{0.5(1.225kg/m^3)(\frac{\pi(82.5m)^2}{4})(11.5m/s)^3)} = 0.30$$

V. CFD ANALYSIS

The analysis is based on following assumptions:

1] Flow is homogenous, steady and incompressible.
2] The thrust is uniform across the rotor area.
3]The static pressure far upstream and downstream of rotor is equal to undisturbed ambient pressure.

After designing the blade, a mesh is created around the blade and the Fluent solver is then used to find the aerodynamics loading on the blade, the fluid streamlines and the torque generated. The standard conditions are used as air at 15℃. Its density is 1.225 kg/m^3 and its viscosity is 1.7894e-05 kg/(m*s), the pressure of 1atm in order to validate the simulations.Using periodicity, simulated the flow around one blade and extrapolate the solution to two more blades in order to visualize the results for a 3-blade rotor. Following figure 3 shows the mesh generation diagram in ANSYS 16.0.

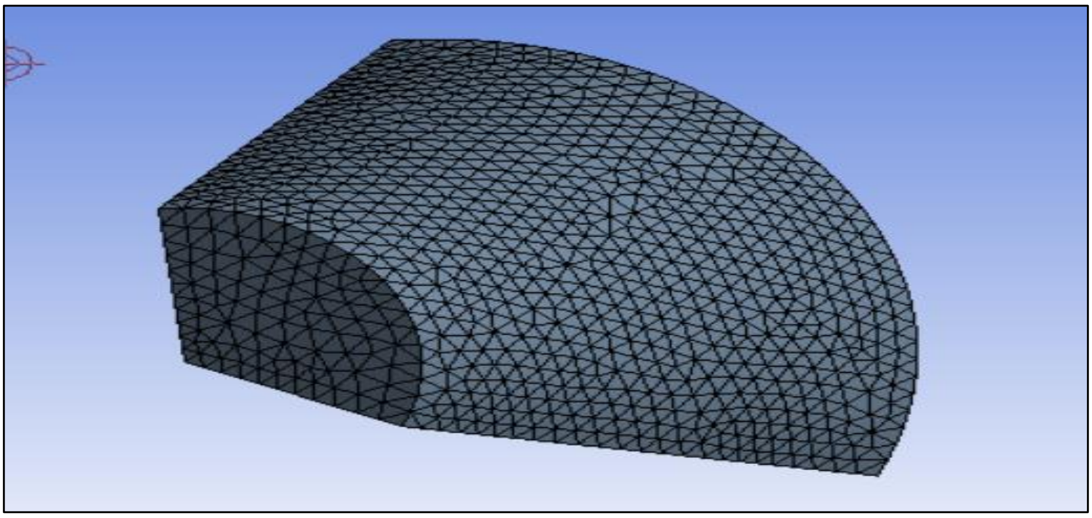

Fig.3 Mesh generation

VI. RESULTS IN FLUENT

Following are the results of analysis:
Figure 4 shows the velocity contour having maximum velocity of 98.05 m/s.

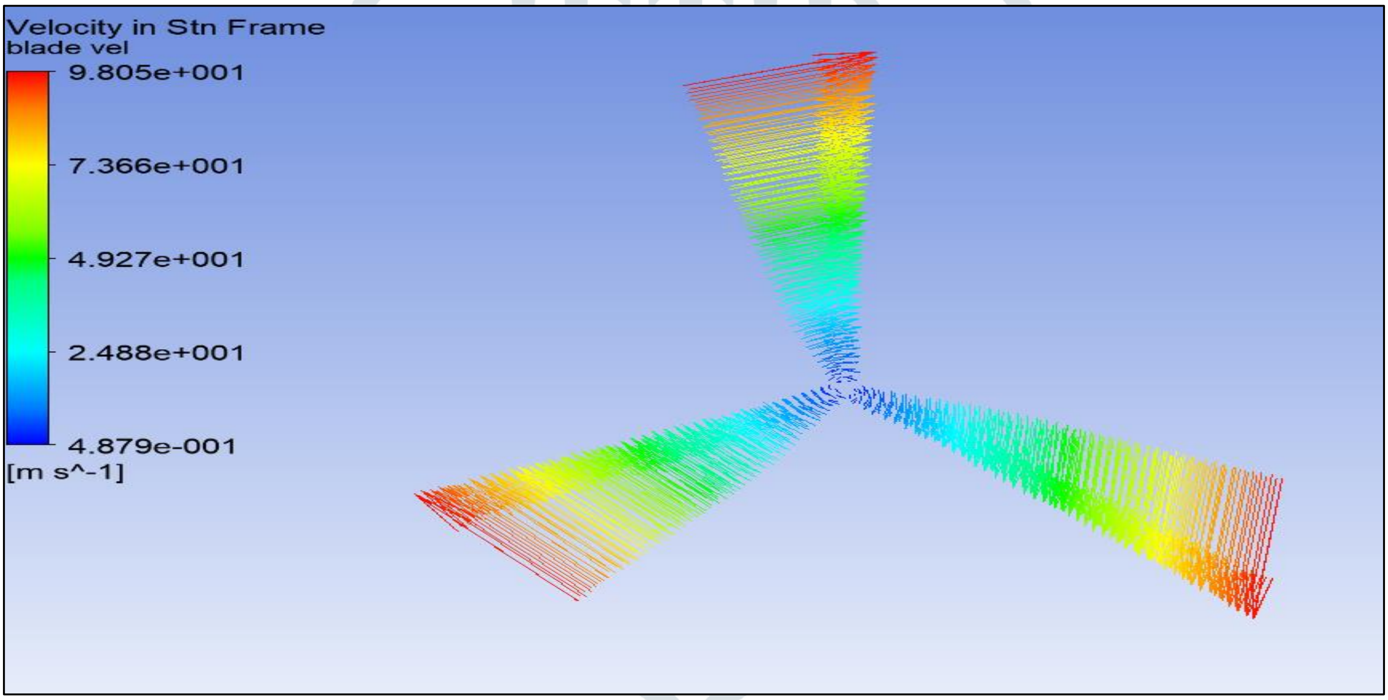

Fig.4 velocity contour

Another essential condition to verify the results is whether a sufficient number of iterations were performed in obtaining the solution (i.e. if the solution converged). The figure below shows how the solution behaves after 3000 iterations. The residuals do not change much between 1500 and 3000 iterations. Therefore 3000 iterations were performed.

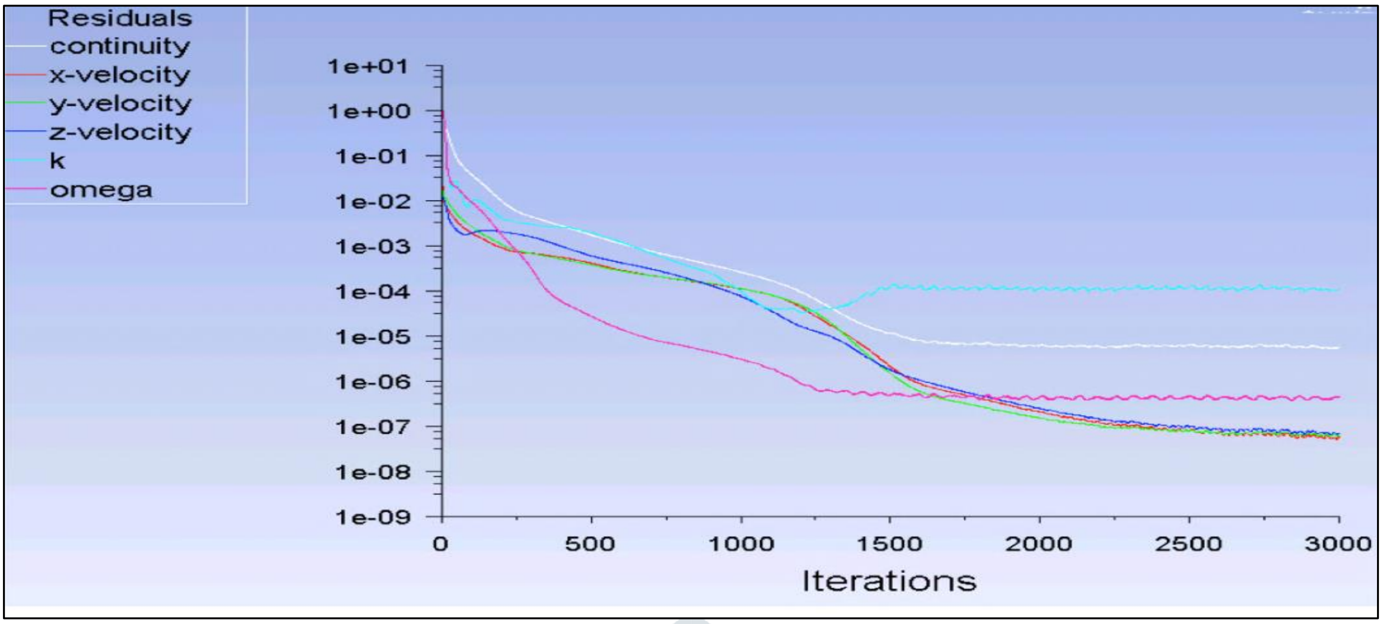

Fig.5 Iteration check

Finally, it is crucial to perform a mesh refinement study. A finer mesh can help achieve a more precise solution of the model but is more computationally expensive. The following figure 6 demonstrates how the results change with a greater number of cells. It is quite clear that the mesh created (which has around 350,000 cells) is not fine enough to obtain a sufficiently accurate solution.

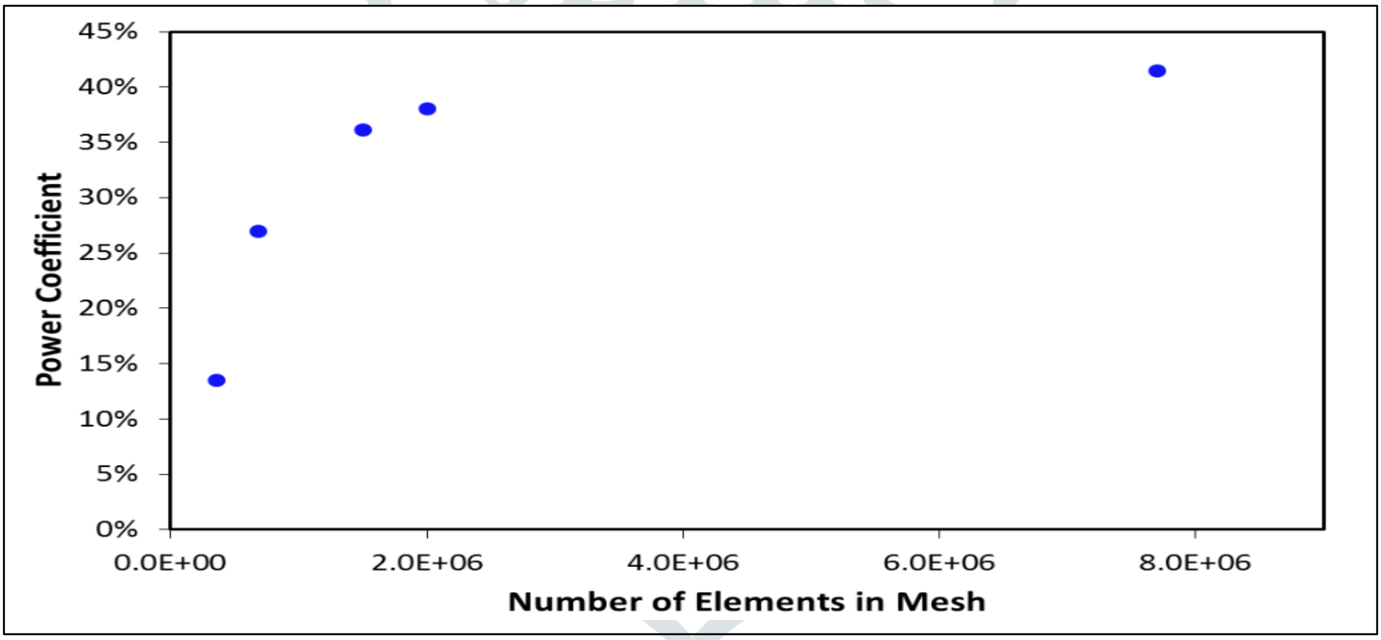

Fig.6 Convergence plot

Convergence of the numerical solution is represented by a negligible change in power coefficient from an approximately four-fold increase in number of elements in the mesh. The mesh was refined by adjusting global and local mesh controls: relevance center, face sizing, inflation and sphere of influence.

VII. CONCLUSION

In this study, the aerodynamic analysis of horizontal wind turbine blade is shown by designing the blade, generating mesh and using fluent solver to determine aerodynamic loading on blade. The blade tip velocity was found to be 98.05 m/s in CFD analysis. This is basically identical to result obtain from calculations which was 98.12 m/s., also the power coefficient is around 0.3. As seen on the convergence plot above, the numerical results showthis value considering a sufficient number of elements in the mesh. For example, using a mesh of 2 million cells, the power coefficient becomes 38% as opposed to the 30% from hand-calculations. These match up quite well considering the many assumptions used in the simple 1D momentum theory. And lastly, the powercoefficient must lie under the Betz limit of 16/27=59.2% for a non-shrouded rotor. The numerical results correctly fall below this limit.

VIII. REFERENCES

[1] Hau, E. Wind Turbines, Fundamentals, Technologies, Application, Economics, 2nd ed.; Springer: Berlin, Germany, 2006.

[2] Dominy, R.; Lunt, P.; Bickerdyke, A.; Dominy, J. Self-starting capability of a darrieus turbine. Proc. Inst. Mech. Eng. Part A J. Power Energy 2007, 221, 111–120.

[3] Holdsworth, B. Green Light for Unique NOVA Offshore Wind Turbine, 2009. Available online: http://www.reinforcedplastics.com (accessed on 8 May 2012).

[4] Viorel Bostan, Ion Bostan, Valeriu Dulgheru, Study of the Aerodynamics of Wind Rotors With Horizontal Axis: Design, Manufacture, Testing,international conference on electromechanical and power systems,2017.

[5] Mohammadi, M. R., Mohammadi, A., Mohammadi, M., NeisiMinaei, H. (2016) Optimization of Small-Scale Wind Turbine Blades for Low Speed Conditions. Journal of Clean Energy Technologies (JOCET), 4, (3), 140-143.

[6] V. Bostan,” Wind turbine”. Patent MD 4213, BOPI nr.3/2013.