International Journal of Engineering and Technical Research (IJETR)
ISSN: 2321-0869 (O) 2454-4698 (P), Volume-3, Issue-7, July 2015

Prediction of Performance Parameters and Determination of Aerodynamic Characteristics of Wind Turbine Airfoil Using CFD Modeling: A Case Study of Adama I Wind Turbine Blade Airfoil Section

Million Merid Afessa, Dr. - Ing. Abebayehu Assefa

***Abstract*— In the current situation of global energy crisis, energy derived from renewable resources has grown significant attention. Among, wind energy is a very interesting. In wind energy, wind turbine technology and its aerodynamic characteristics of the airfoil forming the blade is important. Thus predicting performance parameters and determining aerodynamic characteristics are essential. However, these require experimental wind tunnel and validation tools such as CFD.**

The main objective of this study is focused on predicting aerodynamic characteristics of an airfoil. Simulation was done to deduce aerodynamic parameters: lift, drag, lift to drag ratio, contour plot of velocity and pressure distribution. This can reduce dependence on wind tunnel testing. The simulation was done on airflow over a 2D NACA 63-415 airfoil using FLUENT (version 6.3.26) at various AOAs (-5^0 to 20^0) using two turbulence models (S-A and SST k- ω) with the aim of selecting the most suitable model. Domain discretization was carried out using structured quadrilateral grid generated with GAMBIT (version 2.3.16).

Comparisons and validation were made with available experimental data. Accordingly, it was recorded that the two turbulence models achieved a reasonable and good agreement in predicting the coefficients. Among the models, studied the most appropriate turbulence model were the two equation models, which had good agreement with the experimental data than S-A one equation model. As a result, it was decided to use the SST k- ω turbulence model for the main analysis with acceptable deviations in results of 9.028% for lift and 12.203 % for drag coefficients.

***Index Terms* — Airfoil, Angles of attack (AOAs), Computational Fluid Dynamics (CFD), and SST k- ω.**

I. INTRODUCTION

Global demands for renewable energy resources have been exponentially increasing in the 21st century. This is due to continuous expansion of industrial development, depletion of fossil fuels and emerging environmental consciousness. Above all, increasing in energy demand and growing recognition of global warming and environmental pollution, has become a cross cutting topic for many countries of the world. However, to respond to the energy demand and environmental problem, more and more countries have prioritized renewable energy sources such as wind, solar, hydropower, biomass, geothermal, etc., as replacements for nonrenewable sources such as, fossil fuels **[1].**

As compared to other renewable energy sources, wind has long been glorified as the ultimate solution to the world's energy and environmental problems. Thus, as the most promising renewable energy source and believed to play a critical role in global power supply in the 21st century [1]. As a result, wind energy prices in the 21st century have become roughly comparable in many cases against conventional forms of electricity generation **[2].**

According a report received from Global Wind Energy Council (GWEC) the total amount of wind energy installed has reached 282.5 GW by the end of 2012 around the world. Moreover, its fast growth ensures that the wind power will be an important part of electricity generation in the close future according to the annual market report of Global Wind Energy Council (GWEC) **[3].**

However, recent wind power developments in east Africa with a total of 171 MW (Adama I 51 MW and Ashegoda 120 MW) projects completed and 153 MW Adama II wind project under development in Ethiopia and a 300 MW project under development in Kenya are to be mentioned. Hopefully, these early projects will make a substantial contribution to the total generating capacity in each of these countries. If successful, they will herald a much broader uptake of wind on the continent in the coming few years **[4].**

Generally, wind energy could meet global demand yet only provides small fraction of power consumption [5]. The wind turbine developed to capture wind power and use it for generating electricity is the technology behind and one of the fastest growing industries within the rural and urban areas of countries all around the world **[5].**

Horizontal axis wind turbine (HAWT) is the least expensive and clean way to harness this important energy source [6]. Thus, it is important to note that prediction of performance parameters and determination of aerodynamic characteristics such as lift and drag on a HAWT blade airfoil is an

Million Merid Afessa , School of Mechanical Engineering , Jimma Institute of Technology /JiT/ Jimma University, Jimma, Ethiopia , Mobile No.+251913205174.
Dr.-Ing. Abebayehu Assefa , Associate Professor in Thermal Engineering School of Mechanical and Industrial Engineering , Addis Ababa Institute of Technology /AAiT / Addis Ababa University, Addis Ababa , Ethiopia , Mobile No., +251911 221876.

indispensable, but a complex process **[6].** Accordingly, aerodynamic performance parameters of wind turbine blade airfoils can be determine using computational fluid dynamics (CFD), which is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems of fluid flows **[7], [8].**

Therefore, this study addresses issues that are related to prediction of performance parameters and determination of aerodynamic properties; 2D CFD simulations of HAWT airfoil section are conducted to give a better understanding of flow physics. In addition, comparison analyses have been done to validate results with experimental data from literature **[9]** and simulation are done to verify the reliability and accuracy of CFD model.

II. Problem Description

Despite of plentiful wind resources in Ethiopia the amount utilized to date for power generation is very small and insignificant. It is a well-known fact that growing economy of the country demands sufficient and reliable supply of electric energy and this demand could be fulfilled by increasing the power generation and broadening its capacity. Accordingly, design and development of more efficient and reliable wind turbine is critically significant.

.

In the design and development of wind turbine, it is very important that an accurate assessment is made on the aerodynamic characteristics of the airfoils. In the past, this was determined using prototypes and testing their performance in wind tunnels. This required high capital and running cost of labs, skill man power in manufacturing prototypes accurately and in realization of data, interpretation of result etc.

However, modern wind turbine airfoil design process is improved with the implementation of Computational Fluid Dynamics (CFD) software's. CFD enables the analysis of fluid flow through very complex geometry and boundary conditions. Therefore, if airfoil characteristics cannot be determined by experimental tests, one will rely on computational fluid dynamics (CFD) methods to minimize the problem related with manufacture of prototypes and time spent in wind tunnel or test labs.

III. Previous Numerical Studies

With intensive progress of computer technology in recent time, Computational Fluid Dynamics in the form of steady RANS has made significant improvements in the prediction of airfoil performance. Some of related studies are discussed below.

According to the investigation done by **L. X. Zhang et al., (2013)** numerical simulation had become an attractive method to carry out researches on structure design and aerodynamic performance prediction of wind turbines, while the prediction accuracy was the major concern of CFD. The main objectives of the simulation was to develop a two - dimensional CFD model, at the same time a series of systematic investigations were conducted to analyze the effects of computational domain, grid number, near-wall grid and time step on prediction accuracy. Then efforts were devoted into prediction and analysis of the overall flow field, dynamic performance of blades and its aerodynamic forces. The final out came of the results well agrees with experimental data. It demonstrates that RNG k-ε turbulent model is great to predict the tendency of aerodynamic forces **[10].**

Eleni et al., (2012) performed the analysis of a two dimensional subsonic flow over a NACA 0012 airfoil at various angles of attack and operating at a Reynolds number of 3×10^6. The flow was obtained by solving the steady-state governing equations of continuity and momentum conservation combined with one of the three turbulence models [Spalart-Allmaras, Realizable and shear stress transport (SST)]. The aim of the work was to validate the models through the comparison of the predictions and the free field experimental measurements for the selected airfoil and to show the behavior of the airfoil and to establish a verified solution method **[11].**

Kandwal et al., (2012) investigated an inviscid flow over an airfoil surface using CFD. The major objective was to deduce the lift and drag properties using computational methods, aimed to reduce the dependency on wind tunnel testing. The study is done on air flow over a 2D NACA 4412 airfoil using ANSYS FLUENT (version 12.0.16), to obtain the surface pressure distribution, from which drag and lift were calculated using integral equations of pressure over finite surface areas. The drag and lift forces can be determined through experiments using wind tunnel testing. The CFD simulation results show close agreement with those of the experiments, thus suggesting CFD is a reliable alternative to experimental method in determining drag and lift coefficients **[12].**

Saraf et al., (2011) developed a procedure to numerically model airflow over airfoils using GAMBIT and FLUENT. This model presented the analysis of two - dimensional subsonic flow over NACA 4412 airfoil at various angles of attack and operating at a velocity of 50 m/s. The flow was obtained by solving the steady-state governing equations of continuity and momentum conservation combined with one of the two turbulence models: Spalart Allmaras and k-ω standard. The main attention was to validate those models through the comparison of the predicted results and the free field experimental measurements. At the same time the work was to show the behavior of the airfoil and to establish a verified solution method. The calculations show that the turbulence models used in commercial CFD codes do not give accurate results at high angles of attack **[13].**

Rajendran (2011) focused on the potential of an incompressible Navier–Stokes CFD method for aerodynamic performance analysis. The main objective was to put side by side the CFD results obtained from the simulation and validated against experimental data of the NREL power performance testing activities. Comparisons were shown for the surface pressure distributions at several stations along the blades as well as for the field domain. The simulation result obtained suggest that the method can cope well with flows encountered around horizontal axis wind turbines providing useful results for their aerodynamic performance in the meantime revealing flow details near and off the blades **[14].**

Wolfe (1997) is the other researcher who worked on the capabilities and accuracy of a representative CFD code to predict the flow field and aerodynamic characteristics of typical HAWT airfoils. He made comparisons of the computed pressure and aerodynamic coefficients with wind tunnel data. Like other **[11], [13],** the work indicated two areas in CFD that require further investigation: transition prediction and turbulence modeling. The result of the studies shows that the laminar-to-turbulent transition point must be modeled correctly to get accurate simulations for attached flow. Calculations also show that the standard turbulence model used in most commercial CFD codes, the k-ε model, is not appropriate at angles of attack with flow separation **[15].**

IV. 2-D CFD Modeling Process Flowchart

There are practiced ways and hints in many literatures and these were followed for guidance. Fig1 shows road map when using CFD to reproduce airfoil characteristics.

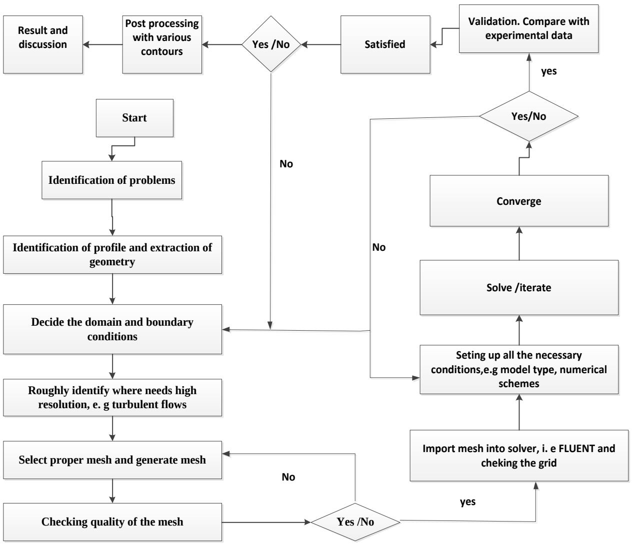

Fig1: Flowchart of typical 2D CFD modeling process

V. Mathematical Formulation

The governing equations of aerodynamics such as the continuity, momentum and energy equations are highly nonlinear, partial differential, or integral equations. Accurate analytical solution to these equations does not exist. CFD approach facilitates a solution to the governing equations [16]. The equation for conservation of mass or continuity equation can be written as follows **[13], [17].**

$$\frac{d\rho}{dt} + \nabla.(\rho\vec{u}) = s_m \qquad (1)$$

This is valid for both incompressible as well as compressible flows. The source S_m is the mass added to the continuous phase from the dispersed second phase and which is user-defined sources **[13].** Conservation of momentum equation is described as:-

$$\frac{\partial(\varrho\vec{u})}{\partial x} + \nabla.(\varrho\vec{u}\vec{u}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F} \qquad (2)$$

In the equation p is the static pressure **τ** is the stress tensor, **g** is the gravitational body force and F is external body force which arises from interaction with the dispersed phase, respectively. The equation also contains other model-dependent source terms such as porous-media and user-defined sources **[6].** The stress tensor τ is given by:

$$\bar{\bar{\tau}} = \mu\left[(\nabla\vec{u}) + \nabla\vec{u}^{T} - \frac{2}{3}\right]\nabla.\vec{u}I \qquad (3)$$

In the above equation μ is the molecular viscosity, I is the unit tensor, and the second term on the right hand side is the effect of volume dilation **[13].**

For the 2-D, steady and incompressible flow the continuity equation is

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad (4)$$

x and y directions momentum equations for viscous flow are given below, respectively

$$\frac{\rho Du}{Dt} = -\frac{\partial P}{\partial x} + \frac{\partial\tau_{xx}}{\partial x} + \frac{\partial\tau_{yx}}{\partial y} + \rho f_x \qquad (5)$$

$$\frac{\rho Dv}{Dt} = -\frac{\partial P}{\partial y} + \frac{\partial\tau_{xy}}{\partial x} + \frac{\partial\tau_{yy}}{\partial y} + \rho f_y \qquad (6)$$

Since the flows considered have 2-D characteristics the term in the Z direction from the continuity equation $\frac{\partial w}{\partial z}$ and from momentum equation $\frac{\partial\tau_{zx}}{\partial z}$ and $\frac{\partial\tau_{yxy}}{\partial z}$ are neglected [13], [17].

VI. Turbulence Models

To make the most appropriate choice of model for specific problem understanding the capabilities and limitations of the various options are very important **[18] [19].** A number of different turbulence models were suitable candidates for modeling the flow over airfoil surface. In this study, two turbulence models are selected to assess the prediction capability.

Spalart - Allmaras One-Equation Models (SA)

Spalart - Allmaras (SA) turbulence modeling is a one-equation modeling [11], [13], [19]. This solves a single transport equation for a quantity which is used to obtain the turbulent viscosity.

$$\frac{D\tilde{v}}{Dt} = c_{b1}(1 - f_{t2})\tilde{S}\,\tilde{v} + \frac{1}{\sigma}\left[\nabla.\left((v + \tilde{v})\right)\nabla\tilde{v} + c_{b2}(\nabla\tilde{v})^2\right] - \left(c_{w1}f_w - \frac{c_{b1}}{k^2}f_{t2}\right)\left(\frac{\tilde{v}}{d}\right)^2 + \quad f_{wt1}\Delta U^2 \qquad (7)$$

In FLUENT, however, the Spalart-Allmaras model has been implemented to use wall functions when the mesh resolution is not sufficiently fine. This might make the model less sensitive to numerical error **[18].** In terms of computation, the Spalart-Allmaras model is the least expensive turbulence model of the options provided in FLUENT, since only one turbulence transport equation is solved. In the turbulence model of Spalart-Allmaras the transport equation can be written as shown below, in the form of the operating parameter **ṽ** which is referred to as the Spalart-Allmaras variable **[11], [13] ,[19]** . Each variable in the production term is defined as:

$$\tilde{S} \equiv S + \frac{\tilde{v}}{K^2d^2}\left[1 - \left(\frac{\tilde{v}}{v}\right)\left(1 + \frac{(\frac{\tilde{v}}{v})^4}{(\frac{\tilde{v}}{v})^3 + C_{v1}{}^3}\right)^{-1}\right] \qquad (8)$$

$$f_w = \frac{\tilde{v}}{\tilde{S}K^2d^2}\left[1 + c_{w2}\left(\left(\frac{\tilde{v}}{\tilde{S}K^2d^2}\right)^5 - 1\right)\right](1 +$$

$$c_{w3}{}^{6})^{\frac{1}{6}}\left\{\left[1+c_{w2}\left(\left(\frac{\tilde{\nu}}{\tilde{S}K^{2}d^{2}}\right)^{5}-1\right)\right]^{6}+c_{w3}{}^{6}\right\}^{\frac{-1}{6}} \quad (9)$$

$$f_{t1} = C_{t1} g_t \exp\left[-C_{t2}\frac{\omega_t{}^2}{\Delta U^2}\left(d^2+g_t{}^2 d_t{}^2\right)\right] \quad (10)$$

$$f_{t2} = C_{t3}\exp\left[-C_{t4}\left(\frac{\tilde{\nu}}{\nu}\right)^2\right] \quad (11)$$

The empirical constants of the Spalart-Allmaras model available in FLUENT are **[6], [16], [19]:**

$$c_{b1} = 0.1355\,,\sigma = \frac{2}{3}, c_{b2} = 0.622\,, k = 0.4187\,,$$
$$c_{w1} = 3.239\,,\quad c_{w2} = 0.3\,, c_{w3} = 2.0\,,$$
$$C_{v1} = 7.1\,, C_{t1} = 1, C_{t2} = 2\,, C_{t3} = 1.2\,, C_{t4} = 0.5$$

Shear-Stress-Transport k–ω Models (SST k- ω)

Two equation turbulence models are one of the most common types of turbulence models. Models like the k-epsilon and the k-omega model have become industry standard models and are commonly used for most types of engineering problems. This blending makes the k- ω SST model valid for a wide range of flows and is recommended for airfoils **[20].** Along with the Spalart-Allmaras model, two-equation models make up the bulk of the turbulence models used for CFD [11]. The turbulence kinetic energy, k, and the specific dissipation rate, ω, are obtained from the following transport equations **[6], [11] ,[13].**

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x}(\rho k u_i) = \frac{\partial}{\partial x_i}\left(\Gamma_K \frac{\partial k}{\partial x_i}\right) + G_K - Y_K + S_K \quad (12)$$

$$\frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x}(\rho\omega u_i) = \frac{\partial}{\partial x_i}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_i}\right) + G_\omega - Y_\omega + S_\omega \quad (13)$$

In the above expression, G_k represents the generation of turbulence kinetic energy due to mean velocity gradients. G_ω represents the generation of ω. Γ_ω and Γ_k represent the effective diffusivity of ω and k, respectively. Y_k and Y_ω represent the dissipation of k and ω due to turbulence. S_k and S_ω are user-defined source terms**.** All of the above terms are calculated and described in reference **[6], [19], [21].**

VII. 2-D AIRFOIL MODELING

The first step in the process of airfoil modeling is to import the points that make up the upper and lower surface of the airfoil. The points are created from a set of coordinate points. The coordinate points file for NACA airfoils are imported from UIUC airfoil coordinate database as dat. file **[22].**

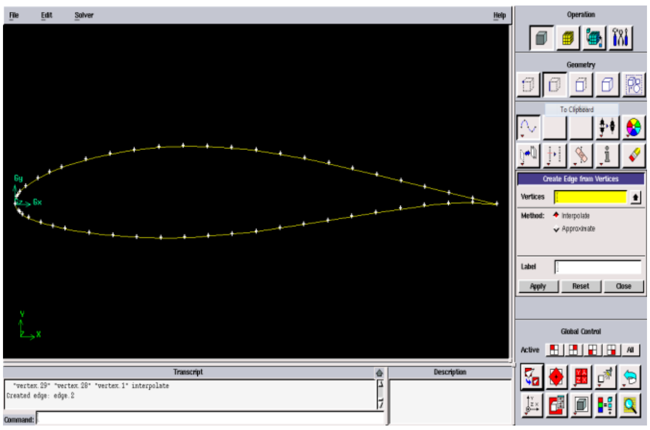

Fig2: NACA 63-415 airfoil geometry developed using GAMBIT

In an external flow such as over an airfoil, it is needed to define a far-field boundary and mesh the region between the airfoil geometry and the far-field boundary. The flow is 2D and the computational domain used is a C-type far field in GAMBIT, which is commonly used for structured meshes **[5].** There are recommended values used in several studies in the literatures **[5], [11], [13], [23]**. The final domain constructed on GAMBIT with specified boundary conditions is shown in Fig3 below.

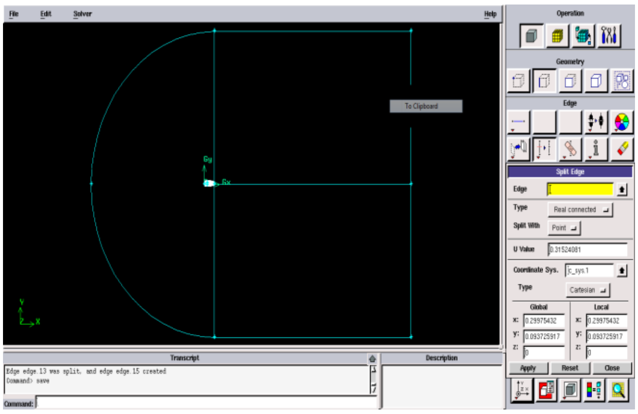

Fig3: Far field boundary and domain developed on GAMBIT

In order to save the computational space and time available for the simulation, this study utilized structured quadrilateral mesh type. Model grid distributions for all faces and close to the airfoil are given in Fig4 and Fig5 respectively.

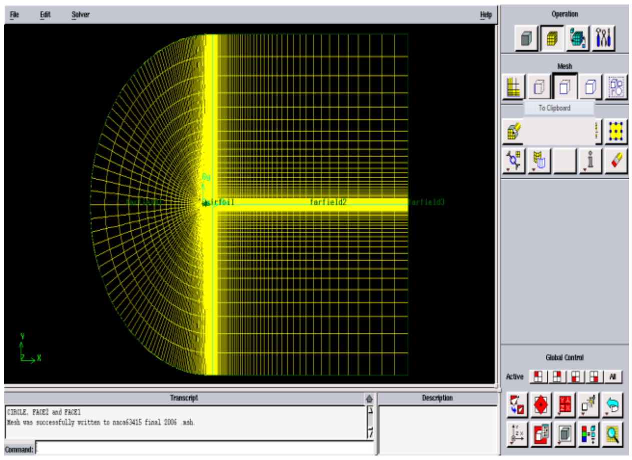

Fig4: Domain meshing over the system using GAMBIT

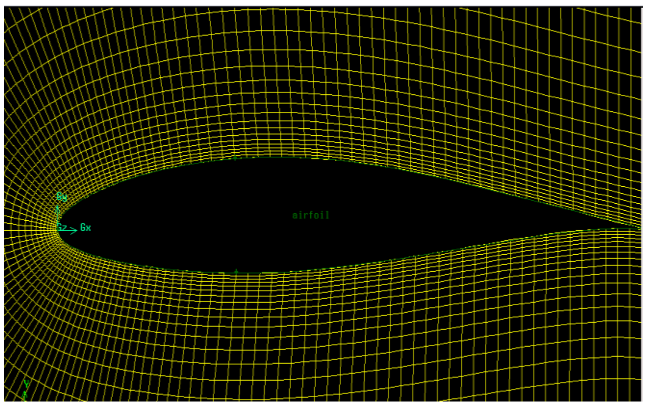

Fig5: Mesh around NACA 63-415 airfoil

Solution Methods

Once a good quality mesh has been generated throughout the entire domain and around the airfoil, the mesh file is imported into the CFD solver, FLUENT. Before the governing equations could be solved, the appropriate settings are first enabled throughout the solver interface and the correct boundary conditions are specified to accurately match the conditions in which the calculations are performed. To more accurately match the experimental conditions, the simulation is performed under the same conditions with the experimental data.

A CFD solver generally requires setting boundary conditions, defining fluid properties, performing simulation and post processing the results. FLUENT 6.3.26, 2ddp (two-dimensional with double precision) flow solver is employed, which implements 2-D Reynolds averaged Navier-Stokes (RANS) equations using a finite volume-solver. The numerical method employed is the pressure-based solver (PBS). In order to solve the set of governing equations, several iterations of the solution must be performed before a converged solution is obtained.

Fluid flow over the surface of the airfoil is not always aligned with the grid. Hence, the second-order-upwind interpolation schemes for the convection terms are used for all nodes. FLUENT recommends this second order scheme to decrease the numerical diffusion error that is more likely to occur with a first order scheme **[18], [19].** The computational expense of second order methods is more expensive per grid point than first order schemes, but the computational accuracy per overall cost of this higher order method is much greater. Stopping criteria for iterative calculations is based on monitoring both the residual history and the lift and drag coefficients. The convergence of the segregated solution is achieved when the sum of the absolute differences of the solution variables between two successive iterations falls below a pre-specified small number, which is chosen as 10−6 in this study. Fig6 represents a convergence history of NACA 63-415 airfoil for 5^0 AOA.

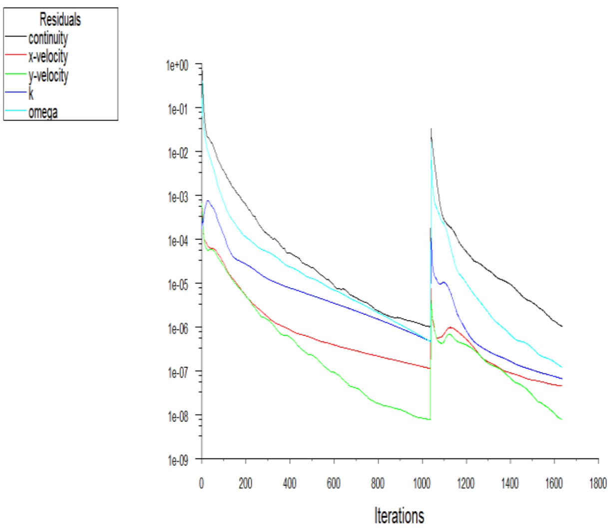

Fig6: Residuals plot with convergence criterion set to 10−6

The working fluid for the simulation is air with density equal to the reference value in the experimental data which is 1.225 kg /m3 **[36]**. A maximum wind speed profile of 40 m/s is taken at the entrance of the domain as boundary condition with fixed turbulence intensity of 1% and turbulence viscosity of 1.789 x10-5 kg / m .s is considered and the Reynolds number for the simulation is about 1.6 $x10^6$, Operating pressure is taken as 101325 Pa. Far-field in the backside of the airfoil is set as pressure outlet with a gauge pressure of zero **[9].**

VIII. Experimental Comparison: 2-D CFD Validation

In this section, the computational results and available experimental measurement data are compared for NACA 63-415 airfoil. The validation process is performed to show the level of accuracy of the CFD code and the computational model and methodologies provided accurate results.

The NACA 63-415 is chosen for the initial validation for it has widely available experimental data to compare with computational results. This is accomplished by plotting the performance parameters, the lift, drag and lift to drag coefficients of the airfoil with AOAs. These data which are available experimentally are conveniently reproduced to numerical data. The experiment was carried out in the VELUX wind tunnel at the Technical University of Denmark and RISO National Laboratory; the tested airfoils were manufactured by LM Glasfiber A, Denmark which is found in reference **[9]**. The most significant results are discussed and presented in graphical form for selected values of angle of attacks.

Fig7 and Fig8 show polar curves of experimental measurement data and computed results of lift and drag coefficients verses AOA, respectively. The results obtained from both turbulence models are in good agreement with experimental values in the simulation of lift coefficient. Fig7 shows the lift coefficient comparison of experimental measurement values with the numerical simulation results for both S-A and SST k- ω models.

The predictive capabilities of the models are compared with maximum percentage errors. The error lies within maximum of 12% in the S-A model as compared to the experimental values, whilst maximum of 9% in the SST k- ω for the calculation of lift coefficients. However, the corresponding errors for the drag show 16 % for the S-A model and 12 % for the SST k- ω model. As many literatures reported **[24],** this deviation is expected in the simulation of drag coefficients because in the actual airfoil the flow is mostly dominated by laminar flow. The turbulence models in the solver S-A and SST k- ω however, assume the flow as fully turbulent, which results in higher drag prediction.

IX. Discussion

Comparison of the results of the turbulence models S-A and SST k- ω with experimental measurement, the SST k- ω model most accurately determines the lift and drag characteristics of the airfoil, NACA 63-415. Fig7 also shows how the S-A model and SST k- ω model both give fairly similar trend for the lift coefficient. With regards to the drag coefficient Fig8 clearly shows that the SST k- ω model also gives a better match to the experimental measurement.

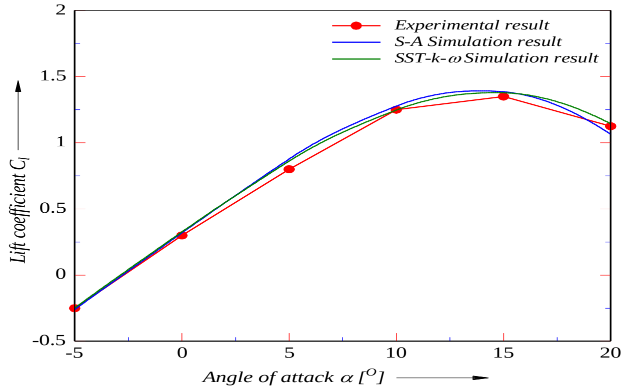

Fig7: Comparison of experimental and computed results for lift coefficient

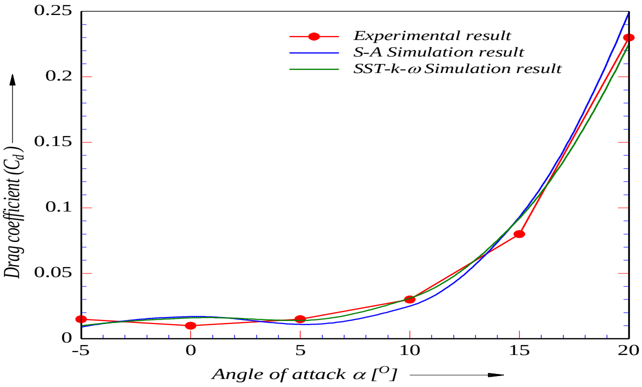

Fig8: Comparison of experimental and computed results for drag coefficient.

The other way of validating the numerical result is plotting the lift to drag (Cl/Cd) ratio, which is shown in Fig9. This ratio increases from zero at around -3^0 to a maximum value at a moderate AOA and then decreases relatively as the angle of attack is further increased. The maximum Cl/Cd value of around 5^0 is derived from the lift and drag characteristics as shown in the Fig9. This is the optimal relationship between lift and drag and the most efficient lift generation for an airfoil is possible. Here, the maximum Cl/Cd ratio is around 54.203 in S-A model while it is around 55 in the SST k- ω model and this value is around 53 in the experimental measurement case.

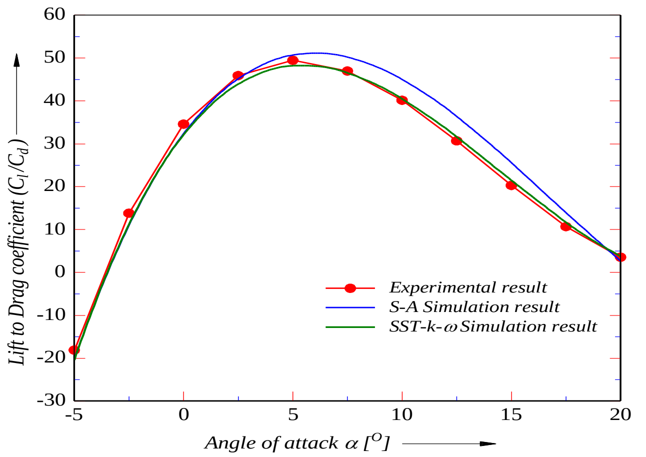

Fig9: Comparison of experimental and CFD result of lift to drag ratio

The deviation of the experimental values and accuracy of the computed lift and drag coefficients are validated against the corresponding experimental values and verified in terms of their percentage deviation. The majority of the deviations are considerably less than 12 % in both S-A and SST k- ω models in the calculation of lift coefficient. However, some important discrepancies are also observed in the calculation of drag coefficient. In the SST k- ω model the maximum deviation observed is less than 12% which is less than 16 % in the S-A model. These differences between the results are likely a result of the turbulence model used in the solver. The other factor responsible for the discrepancy for the result is probably the flow region which is not properly resolved with the RANS approximation, inappropriate boundary layer mesh. Fig10 and Fig11 show the maximum percentage deviation of the models in the calculation of lift and drag coefficients respectively.

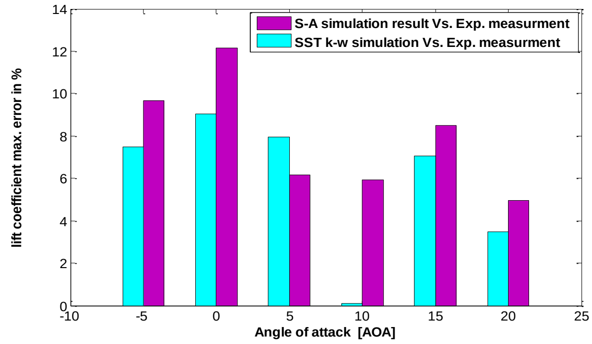

Fig10: S-A and SST k- ω model with experimental result max. Error in % for lift

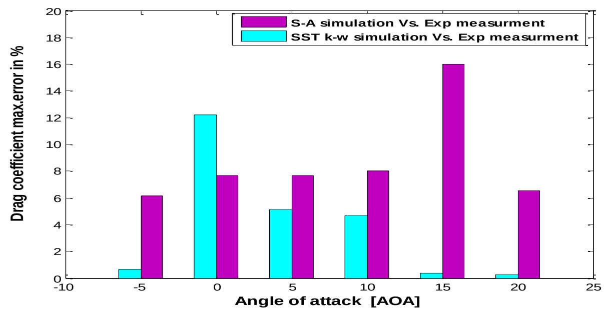

Fig11: S-A and SST k- model with experimental result max. Error in % for drag

In general, both turbulence models had good agreement with experimental data at angle of attack even at small negative angles; the same trend of curve was observed at all angle of attack. However, small discrepancy was observed between the CFD curve and the experiment. It is a well-known fact that lift coefficients are reduced and drag coefficients increased as angle of attack increases. The discrepancy in the drag coefficient, obtained from simulation is due to the fact that the flow over the actual airfoil is mostly dominated by laminar flow. However, the turbulent models S-A and SST k- ω is assumed the flow as fully turbulent over the surface of the airfoil and does not provide a transition from laminar to turbulent flow. This assumption has the potential of causing some error between the simulation result and experimental data. A turbulent boundary plays a substantial effect in the prediction of drag coefficients, thus the error between the simulation and the experiment can be noticed in the comparison of drag coefficient.

In conclusion, by comparing the simulation results and available experimental data for the force coefficients (lift and drag) values through Fig7 and Fig8, it can be concluded that the CFD solution is providing sufficiently accurate results for a majority of the AOAs. However, the most well agreed model was the SST k- ω model. As a result, the SST k- ω turbulence model has been chosen for the main analysis of the case study discuss below.

Case Study of Adama I Wind Turbine Airfoil

The case study discussed here is to predict aerodynamic performance parameters of wind turbine airfoil for which all the major technical specification are given in Table 1 below

Table 1: Technical specifications for GW 77 /1.5 wind turbine **[25]**

Parameters	**GW77**	
	IEC class	**IIA**
Operational parameters	Rated power	1500kW
	Cut in wind speed	3m/s
	Rated wind speed	11.1 m/s
	Cut out wind speed(10min)	22m/s
	Survival wind speed (3min)	59.5m/s
	Design life time	≥ 20 years
	Operating ambient temperature	-30° C to +40° C
	Stand by ambient temperature	-40° C to +50° C
Rotor	Nominal diameter	77m
	Number of blades	3
	Blade airfoil type	LM 37.3P or similar
	Aerodynamic profile, modified NACA and others	NACA 63-4xx and fx 77/79

Project Description

The physical model taken for this numerical calculation is the Adama wind farm project which has capacity of generating 51 MW, 3 blades and 34 sets of 1500kW GOLDWIND (GW) horizontal axis upwind wind turbines , located in the middle part of Ethiopia, about 3.91 km North West of Nazareth. The project was constructed and has started operation in year 2011/ 2012.

According to the feasibility report of Adama Wind Park the design average wind speed of the site at the WTG hub height of 65m is 9.56 m/s. The calculated long term average temperature is 21.2^0C and the atmospheric pressure of the site is 853.3hPa (85330Pa), the air density of the site at hub height range from 0.959- 0.973 kg/m^3. However, for this simulation 0. 97 kg/m^3 is considered **[25] [26].**

The present study utilized CFD solver; FLUENT to predict the aerodynamic performance parameters while fluid with density of 0.97 kg /m^3 flows over the horizontal axis wind turbine (HAWT) airfoil. The aerodynamic performance parameters of the airfoil for average wind speed of 9.56 m/s (corresponding to Reynolds number of 6.1 x 105 and Mach number of 0 .028) were predicted. The aerodynamic profile of the airfoil from UIUC Airfoil Coordinates Database NACA 63-415 airfoil is considered.

The model used for this case study is the same as the validation described in the above sections all the way. The FLUENT solution set up was also identical, however the only difference in the solution set up is the input parameters such as velocity, temperature, pressure and density. Since there is no experimental data available with the same conditions as the selected site, validation of the CFD results and verification of model for different conditions for the NACA 63-415 is required. The calculations were all done with average speed of 9.56m/s flow with SST k-w two equations turbulence model.

X. Results and Discussion

All the information gathered in terms of the lift and drag characteristics results from the pressure and velocity distributions over the surface of the airfoil at a given set of operating conditions are discussed.

An angle of attack range from -5^0 to 20^0 is selected, with an interval of 5^0 between each simulation. The computed results and the NACA 63-415 findings at the given conditions are compiled.

The plot in Fig12 shows the variation of lift coefficient with AOAs. There is a range of AOA where the lift coefficient varies linearly. The lift coefficient is negative at negative AOA until it achieves the zero AOA and then increases with increasing AOA. The lift coefficient is zero when AOA is at around -3^0. The lift coefficient keeps increasing until it reaches the critical angle of attack, point at which maximum lift coefficient attain. The critical angle of attack is around 15^0 and the maximum lift coefficient recorded at this angle of attack is around 1.3596. A further increase in AOAs beyond these AOA causes a decrease in the lift coefficient.

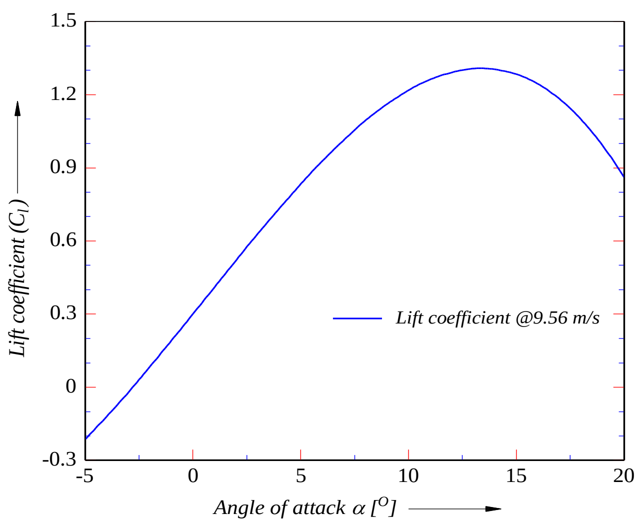

Fig12: CFD result of Cl vs. AOA for - NACA 63-415 @ 9.56 m/s

The values of drag coefficient increase with the angle of attack. However, the drag coefficient values are low for low AOA from -5^0 to 10^0 before it rising exponentially.

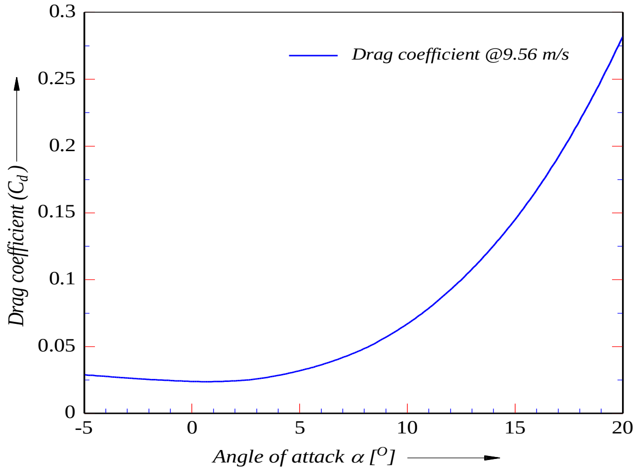

Fig13: CFD result of Cd vs. AOA for - NACA 63-415 @ 9.56 m/s

The maximum lift to drag ratio of an airfoil is another important performance parameter to characterize the airfoil. The optimum angle of attack is where lift to drag (Cl/Cd) coefficient has the maximum value. Fig14 shows the lift to drag ratio plot obtained from the lift and drag data. From the curve the optimum AOA of the airfoil is around 5^0.

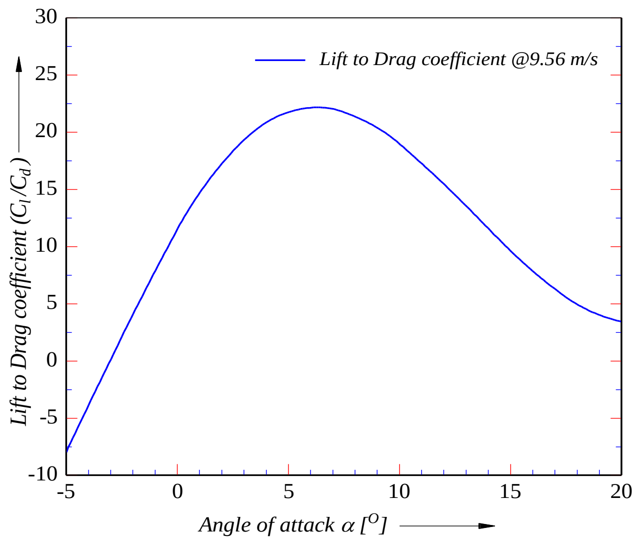

Fig14: CFD result of Cl/Cd vs. AOA for - NACA 63-415 @ 9.56 m/s

Post – processing Results

Fig15 to Fig17 show pressure distribution plots. Fig15 shows the result for 0^0 attack angle and it shows the stagnation point (maximum pressure) at the leading edge of the airfoil. Increasing the attack angle to 5^0, 10^0, 15^0 and 20^0 shows a dramatic change in the pressure distribution. It is evident from the contours plots there is region of very large low pressure on the upper surface of airfoil as compared to the bottom surface of the airfoil. The stagnation point moves to the lower surface as angle of attack increased.

Contour plot of static pressure distribution

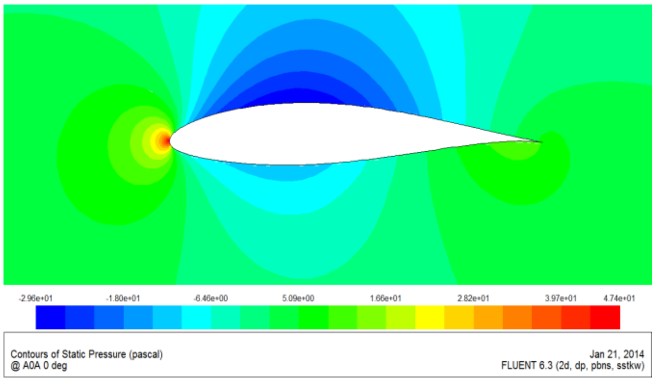

Fig15: Contour plot of static pressure at 0° AOA using SST k- ω turbulence model

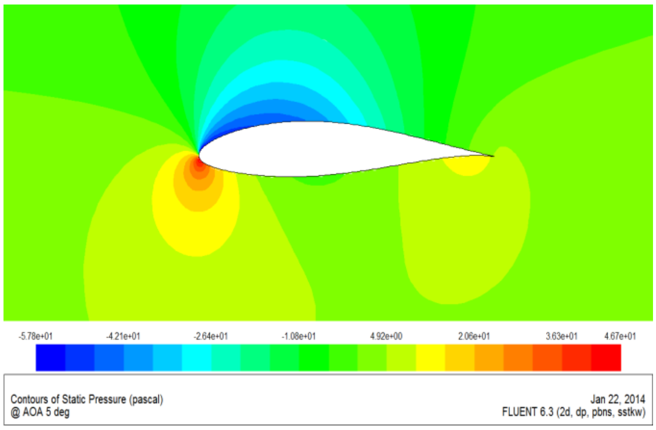

Fig16: Contour of static pressure at 5° AOA with the SST k- ω turbulence model

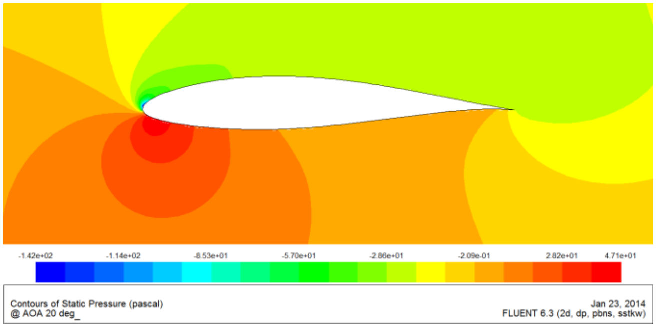

Fig17: Contour of static pressure at 20° AOA with the SST k- ω turbulence model

Contour plot of velocity distribution

The velocities over the upper surface of the airfoil are substantially higher than the lower surface except for negative angle of attack in this case the condition is reversed.

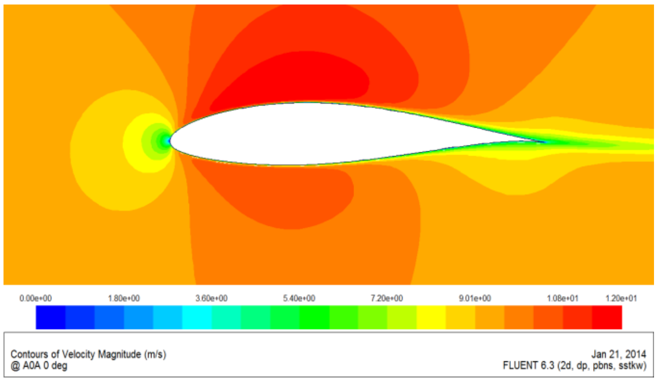

Fig18: Contour of velocity magnitude at 0° AOA with the

SST k- ω turbulence model

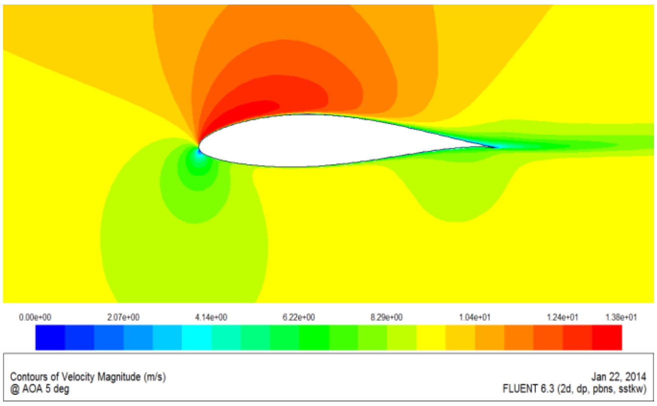

Fig19: Contour of velocity magnitude at 5° AOA with the SST k- ω turbulence model

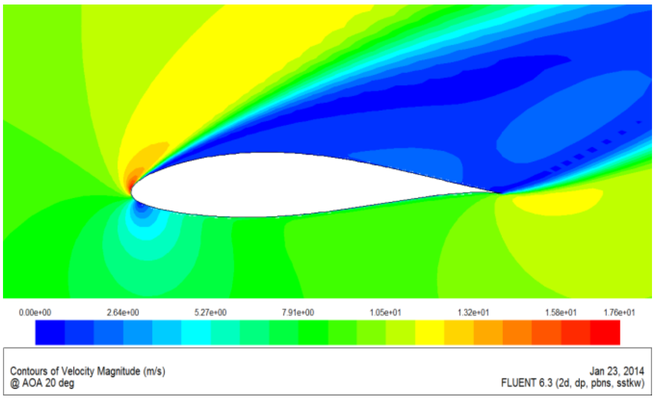

Fig20: Contour of velocity magnitude at 20°AOA with the SST k- ω turbulence model

Contour plot of velocity streamlines

At small angles of attack, the boundary-layers that develop on the upper and lower surfaces of the airfoil do not separate; the flow is attached to the airfoil throughout flow. At large angles of attack, the airfoils develop large adverse pressure gradients that cover the complete upper surface of the airfoil. Separation appears close to the trailing edge and separation point moves upstream with the increase of the angle of attack.

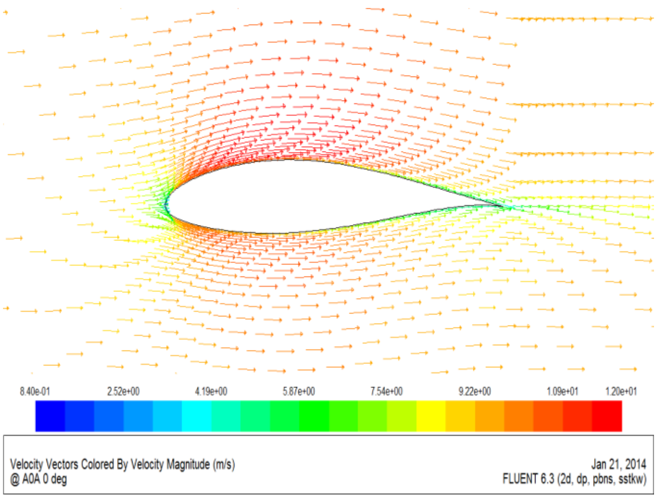

Fig23: Velocity vectors colored by velocity magnitude at 0° AOA with the SST k- ω model

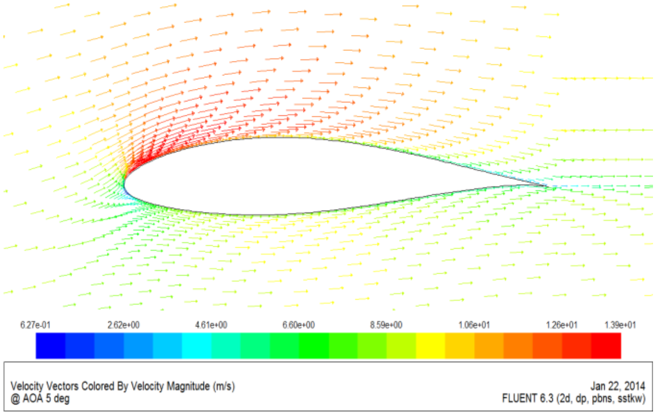

Fig22: Velocity vectors colored by velocity magnitude at 5° AOA with the SST k- ω model

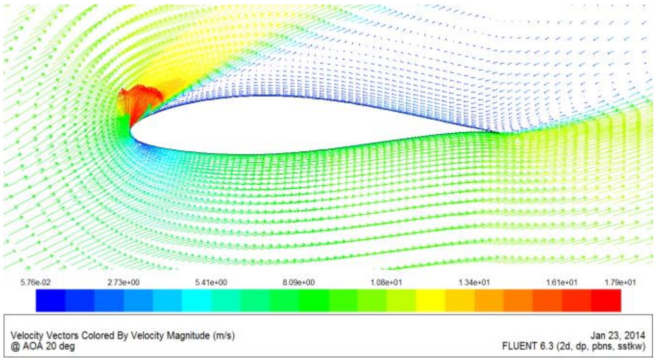

Fig23: Velocity vectors colored by velocity magnitude at 20° AOA with the SST k- ω model

XI. Conclusion

The model has generated results which are in good agreement with experimental data when run in FLUENT under the same conditions. This validates the use of a simpler CFD models for analyzing airflow over airfoils instead of the more expensive and time consuming wind tunnel experiment. However in this study it is found that the ability to successfully obtain specific aerodynamic characteristics is highly dependent on modeling the airfoil section in the preprocessor called GAMBIT and selecting appropriate turbulence model in the solver called FLUENT. Generally, due to limited resources certain issues are not discussed in detailed. This could be considered as an introductory study to illustrate the uses of Computational Fluid Dynamic (CFD) software in the simulation of fluid flow over airfoil surface with different turbulent models. Performing CFD simulation of a complete 3D HAWT is also extended as a future work in this area.

Acknowledgment

My heartfelt thanks go to Dr. _ Ing. Abebayehu Assefa for all the guidance, advice and support he has provided during this study undertaking. I extend my acknowledgment to the Ethiopian Ministry of Education (MOE), Addis Ababa Institute of Technology (AAiT) and Jimma Institute of Technology (JiT), for kindly sponsoring and financing the research. Last but not least, I would like to dearly friends who gave me valuable comments and support during the research undertaking.

ABBREVIATIONS

2ddp	Two –dimensional with double precision
2D	Two dimensional
AOA (α)	Angle of attack
CFD	Computational fluid dynamics
GW	Giga Watts
GWEC	Global wind energy council
HAWT	Horizontal axis wind turbine
kW	Kilo watts
MW	Mega Watts
NACA	National Advisory Committee for Aeronautics
NSE	Navier Stokes Equations
PBS	Pressure- based solver
RANS	Reynolds –Average Navier Stokes
SA	Spalart- Allmaras in turbulence model
SST k−ω	Shear-Stress-Transport k−ω in turbulence

REFERENCES

[1] Wei Tong. (2010). Wind Power Generation and Wind Turbine Design. Radford, Virginia, USA.

[2] Robert Y. Redlinger. (2002). Wind Energy in the 21st Century: Economics, Policy, Technology and the Changing Electricity Industry.

[3] GWEC, (2012).Global Wind Power Cumulative Capacity. [Online] available at: <http://en.wikipedia.org/wiki/File:GlobalWindPowerCumulativeCapacity.png>[accessed may 15, 2012].

[4] Global wind energy outlook. November (2012)

[5] Kay, A. W. (2010), Evaluating the Performance of Commercially Available Computational Codes for Determining the Aerodynamic Characteristics of Wind Turbine Airfoils, University of Strathclyde Engineering.

[6] Ece Sagol, Marcelo Reggio, and Adrian Ilinca,.(2012), Assessment of Two-Equation Turbulence Models and Validation of the Performance Characteristics of an Experimental Wind Turbine by CFD. International Scholarly Research Network, ISRN Mechanical Engineering Volume (2012), Article ID 428671, 10 pages.

[7] Patel, M. B. (2012), A Review on Aerodynamic Analysis of Wind Turbine Blade Using CFD Technique. International Journal of Engineering Research & Technology (IJERT) Vol., 1 Issue 10, ISSN: 2278-0181.

[8] Cao, H. (2011), Aerodynamics Analysis of Small Horizontal Axis Wind Turbine Blades by Using 2D and 3D CFD Modelling. University of central Lancashire.

[9] Christian Bak, Peter Fuglsang, Jeppe Johansen, Ioannis Antoniou. (2000). Wind Tunnel Tests of the NACA 63-415 and a Modified NACA 63-415 Airfoils. Risø National Laboratory, Roskilde, Denmark.

[10] L. X. Zhang, Y. B Lian, X. H. Liu, Q. F. Jiao and J. Guo (2013), Aerodynamic Performance Prediction of Straight-bladed Vertical Axis Wind Turbine Based on CFD. Advances in Mechanical Engineering Volume 2013 (2013), Article ID 905379, 11 pages

[11] Douvi C. Eleni, Tsavalos I. Athanasios and Margaris P. Dionissios, (2012, March). Evaluation of the Turbulence Models for the Simulation of the Flow over a National Advisory Committee for Aeronautics (NACA) 0012 Airfoil. Journal of Mechanical Engineering Research Vol. 4(3), pp. 100-111.

[12] S.Kandwal1, Dr. S. Singh (2012, September). Computational Fluid Dynamics Study of Fluid Flow and Aerodynamic Forces on an Airfoil. International Journal of Engineering Research & Technology (IJERT) Vol. 1 Issue 7, ISSN: 2278-0181.

[13] Amit Kumar Saraf, Manvijay Singh, Ajay Kumar, (2012). Analysis of the Spalart-Allmaras and k-ω standard models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 4412 airfoil. International Journal of Scientific & Engineering Research, Vol 3(8), ISSN 2229-5518.

[14] C. Rajendran, (2010). Aerodynamic Performance Analysis of Horizontal Axis Wind Turbine Using CFD Technique. European Journal of Scientific Research, ISSN 1450-216X Vol.65 No.1 (2011), pp. 28-37.

[15] Walter P. Wolfe, (1997, September). Predicting Aerodynamic Characteristics of Typical Wind Turbine Airfoils Using CFD.SAND96-2345 UC-261.

[16] M. Senthil Kumar and K. Naveen Kumar, (2013). Design and Computational Studies on Plain Flaps. International Journal of Industrial Engineering and Management Science, Vol. 3, No. 2.

[17] Spalart, P. R. and Allmaras, S. R., (1992). A One-Equation Turbulence Model for Aerodynamic Flows. AIAA Paper 92-0439.

[18] Fluent 6.2 User's Guide © Fluent Inc., 2005.

[19] FLUENT 6.3 User's Guide© Fluent Inc. 2006-09-20.

[20] Radermacher, Ryne Derrick,(2012). "Computational Analysis of a Wing Oscillator". Western Michigan University: Available at. http://scholarworks.wmich.edu/masters_theses.

[21] Fernando Villalpando and Marcelo Reggio. Assessment of turbulence models for flow simulation around a wind turbine airfoils. Advances in Computational Fluid Dynamics and Its Applications.

[22] Anon. UIUC Airfoil Coordinate Database. Available from: http://www.ae.illinois.edu/m-selig/ads/coord_database.html.

[23] Sibly School of Mechanical and Aerospace Engineering. FLUENT Tutorials – Flow over an Airfoil. Cornel University. [Online] available at: http://courses.cit.cornell.edu/fluent/index.htm [Accessed June 2013].

[24] Ravi.H.C, Madhukeshwara.N, S.Kumarappa, (2013). Numerical Investigation of Flow Transition for NACA-4412 Airfoil Using Computational Fluid Dynamics. International Journal of Innovative Research in Science, Engineering and Technology Vol. 2, Issue 7, ISSN: 2319-8753.

[25] "Feasibility Study for Ethiopia Adama (Nazret) Wind Park" report. Hydro china Corporation, (2009).

[26] "Proposal for Engineering, procurement and Construction of Adama Wind farm", volume 2 technical offer, HYDROCHINA &CGCOC joint venture November, 2009.

Million Merid , MSc student in Energy Technology at Addis Ababa Institute of Technology , Lecturer at School of Mechanical Engineering ,Jimma Institute of Technology, Jimma University , Jimma , Ethiopia., Member of Ethiopian society of mechanical Engineering [ESME].
Contact addresses: Mobile: + 251913205174

Dr.-Ing. Abebayehu Assefa, Associate Professor in Thermal Engineering, School of Mechanical and Industrial Engineering, Addis Ababa Institute of Technology, Addis Ababa University , Addis Ababa , Ethiopia .
Contact addresses Office: +251118 102176,
Mobile: +251911221876