Numerical Investigation of Drag and Lift Forces on Turbulent Flow on an Airfoil Shaped Body at Different Angles of Attack

Suvanjana Bhattacharyya1*, Shramona Chakraborty2, Sumanta Laha1
1 Mechanical Engineering Department, MCKV Institute of Engineering, Liluah, Howrah, West Bengal, India
2 Computer Science Engineering Department, MCKV Institute of Engineering, Liluah, Howrah, West Bengal, India

Abstract
Numerical analysis of drag and lift force on an airfoil shaped body at different angles is presented. Fundamentals of aircraft design are provided with detailed estimation of the aerodynamic drag and lift of airfoil is presented. This work presents the configuration optimization of an aerofoil body in air using SST computational fluid dynamics (CFD) modeling and compared with the experimental result. The governing equations are solved with a finite-volume-based numerical method. A three-dimensional non uniform grid was generated, in order to critically examine the flow. At different angle the coefficient of drag and lift is determine. Airfoil was placed in a low speed wind tunnel with pressure taps along its surface. The wind tunnel was operated at nominal velocities during the coefficient measurement, a Reynolds number of about 100–2000. The airfoil, with a 10 inch chord, was analyzed at 0°, 5°, 10°, 15°, 20°, 25° angles of attack and with fixed load.

Keywords: CFD, drag force, coefficient of drag, coefficient of lift, wind tunnel

*Author for Correspondence E-mail: suvanjanr@gmail.com

INTRODUCTION
An aircraft during its flight is subjected to aerodynamic forces, namely lift and drag. These forces are primarily produced due to non-uniform pressure distribution on aircraft surfaces and skin friction. The force turns the aircraft structure, which results in the development of stresses and deformation in the structure. The aircraft structural design, therefore, necessitates strength and stiffness analysis for operating flight conditions. This can be accomplished by simulating actual pressure loads on the aircraft and analyzing its structure for strength and deformations. Therefore, to accomplish and verify the structural design one way fluid-solid interaction is sought necessary; where in aerodynamics loads are applied on the vehicle structure for steady-state static structural finite element analysis and its effect on the structure is investigated. The choice of airfoils mainly included NACA series airfoils, NREL S series airfoils, SERI series airfoils, RISF-A series airfoils, FFAW series airfoils [1] and DU series airfoils. At present, the numerical simulation analysis for the performance of wind turbine airfoils [2] mainly concentrated in the influence of mesh density [3], turbulence model [4,5], leading edge roughness [5], airfoil camber [6] and Reynolds number [7] for the aerodynamic performance. Given the reality that the aerodynamic performance analysis of domestic wind turbine machine mainly based on the numerical simulation, the deep study on the simulation for common airfoils would provide reliable reference for the aerodynamic design of wind turbine machine. Substantial knowledge base draws breathe for best execution in the evolution of the wing design for transonic performance in standard large transport aircraft. This demonstration examines the airfoil in a low speed wind tunnel at varying angles of attack. Symmetric airfoils are used in many applications including aircraft vertical stabilizers, submarine fins, rotary and some fixed wings. A 3D wing section (Figures 1 and 2) is analyzed at low speeds for lift and drag characteristics at different angles.
Fig. 1: Internal Arrangement of Spars and Ribs.

Fig. 2: Diagrams of Wind and Force Vectors Acting at Airfoil Sections.
Controlling the flow field around bodies such as cylinders, airfoils, flat plates, compressor and turbine cascades, etc. is of interest in aerodynamics because by controlling the flow field properly, we can modify the flow field structure so that the aerodynamic characteristics of the body would be improved. Two types of mechanisms are followed for controlling the flow field, those are: a) passive flow control methods; b) active flow control methods.

In passive flow control methods, system doesn’t require energy as input. Some examples of this kind of flow control methods are boundary layer trips, roughness elements, ejector nozzles and surface perturbations. A salient feature of this method is it is only useful in low Reynolds number flows because at high Reynolds number, drag coefficient would rise significantly.

In active flow control methods, the input energy of the system can be in forms of heat, laser, electron beams, micro waves, acoustic energy, plasma, etc. Some examples of these methods are surface blowing to contrast a pseudo streamlined-shaped surface, surface suction to obtain a thinner boundary layer, oscillations of the body, synthetic jets. This method is more flexible than the passive ones because the energy in the input of the system can be turned on and off as necessitate. The foremost objective of active flow control methods is to reattach the separated flow to the surface and this method has been applied by many researchers during recent years.

**Fig. 3:** Iso-vorticity Fields using k-ω SST Modelling for 5 and 10 Degrees Angle of Attack Upstroke.

**Fig. 4:** Angle of Attack of the Airfoil Blade.

**TURBULENCE MODELLING**

The prediction of dynamic stall phenomenon at low Reynolds number turbulent flow is a crucial need in aeronautics and more specifically in rotorcraft dynamics. In this context, the forced unsteadiness interacts non linearly with the fine scale, random turbulence and produces a strong irreversibility effect that usually leads to hysteresis loops in the aerodynamics coefficients versus the angle of incidence curves. Under these conditions of non-equilibrium turbulence, standard modeling approaches are often insufficient to predict the dynamic stall at moderate Reynolds...
Numerical Investigation of Drag and Lift Forces

Bhattacharyya et al.

number. The applications of these flows occur in turbo-machinery and in helicopter rotor blades as well as in wind turbine airfoils. It is important to have a good prediction of the dynamic stall to ensure efficiency for the design. Transition SST model is three dimensionally performed. Figure 3 shows the iso-vorticity fields for 06 different angles of incidence. From 0° to 25° upstroke, the flow remains attached to the profile.

PHYSICAL MODEL
Three-dimensional numerical simulation model of airfoil body having non uniform grid was generated, in order to critically examine the flow (Figure 4). In this study, 3-D numerical of airfoil body with 8 inch chord, in order to examine the flow critically a refining of the grid was necessary. The grid generations and numerical simulations are performed using commercial Ansys 15 software package Gambit and Fluent, respectively. Figure 5 presents the mesh generation of the computation domain using hybrid method of size function and boundary layer technique.

MATHEMATICAL FORMULATION
Drag force resists the movement of solid objects through fluid. This drag force is made up of by the components of frictional force and pressure force, which is being integrated to all over the surface of the body. The drag force is very much dependent on shape of the body hence results in complicacy of the formulae of drag force. So the drag force equation is developed using the drag coefficient approach. In this approach the drag force is related to the stagnation pressure $\frac{1}{2} \rho U^2$ and exposed area $AD$. The exposed area is taken as projected area parallel to the fluid flow for convenience. Thus, the relationship is expressed by $FD = CD.AD.\rho.U^2/2$.

A linear mathematical analysis has been carried out in this section to anticipate $CD$ and $CD_\alpha$ along with the prominent features of their dependence on $\alpha$: (i) $CL$ becomes maximum at $\alpha = 25$ degree and then drops to a very small value at $\alpha = 5$ degree, (ii) $CD$ is maximum at $\alpha = 25$, (iii) the inflection point of $CD (\alpha)$ occurs at $\alpha = 10$.

With increase in $\alpha$ from 0 to 25 degree, the area $AL$ of the airfoil projected on the x-z plane shrinks (since $AL = c \cos \alpha$). Thickness affects $AL$ appreciably only at $\alpha = 25$, since $AL$ is directly linked with the magnitude of the lift force.

Similarly, the area $AD$ projected on the y-z plane has been expressed as $AD = csin \alpha$, which is being connected with the magnitude of the drag force.

With increase in $\alpha$ from 0 to 25 degree, the bluffness of the airfoil changes from a streamline to maximum, where bluffness is defined as the body height, i.e., $c \sin \alpha$, projected in the y–z plane. It is plausible to assume that the base pressure ($P_b$), defined as the pressure at the midpoint of the suction surface, increases with $\alpha$ and its increase, i.e., $dP_b$, is directly proportional to the increase in the ratio of bluffness to $c$, i.e., $d\{(c \sin \alpha)/c\}$, viz. $dP_b \alpha d\{(c \sin \alpha)/c\}$.

Fig. 5: Mesh Generation.
\[ L = cC_1C_3\sin\alpha + cC_1C_3\sin\alpha \] (i)
\[ P_b = C_1\sin\alpha + C_2 \] (ii)
\[ L = C_1A_LP_b \] (iii)
\[ D = C_1A_DP_b \] (iv)
where, \( C_1 \) and \( C_2 \) are two constants. \( P_b \) is directly linked with \( CL \) and \( CD \) and has been assumed to be the representative pressure for the entire base surface.

\[ D = cC_1C_3\sin\alpha \sin\alpha + cC_2C_3\sin\alpha \] (v)

The trapezoidal rule has been used several times throughout the analysis in order to numerically carry out required integration of the measured data. The following is the trapezoidal rule (method) also known as the trapezium rule.

\[
\int_I \approx \frac{1}{2} \left[ f(a) + f(b) \right] (b-a) \quad \text{(vi)}
\]

RESULT AND DISCUSSION

Different lift and drag forces are calculated for different shapes and sizes of airfoil therefore, to explore the pros and cons of airfoils, non dimensional lift coefficient and drag coefficient are used. The calculated equations of airfoils’ lift and drag coefficient are shown as follows:

\[ C_l = \frac{L}{0.5\rho V_r^2 l} \]
\[ C_d = \frac{D}{0.5\rho V_r^2 l} \] (vii)

\( L \) is the lift force suffered by airfoil, \( D \) is the drag force suffered by airfoil, \( \rho \) is air density, \( V_r \) is the relative velocity when air flowed through the airfoil and \( l \) is the airfoil’s chord length. Airfoil’s lift drag coefficient curve and the calculated result of different Reynolds number are shown in Figures 6 and 7 and compared with Bhattacharyya, et al. [8] experimental result. It could be seen from the results, the lift and drag coefficient curve of different Reynolds number has consistent movements and shapes with the experimental curve. And for the lift coefficient curve, the calculation results moderate Reynolds number value (Re 1000) was closest with the experimental data. While the drag coefficient curve of four Reynolds number varied greatly with the experimental data, especially one Reynolds number, and other three Reynolds number had smallest difference between the experimental data. Drag coefficient’s larger deviation may be caused by the drag coefficient’s sensitivity to the surface roughness and other factors.
CONCLUSION
With the help of CFD software Fluent, the aerodynamic performance of the dedicated airfoils was simulated numerically when Reynolds number was 100–2000 and the attack angle in calculation changed from 0° to 25°. When the attack angle changed from 0° to 25°, steady numerical methods could be applied to predict the aerodynamic performance of airfoil, the lift and drag coefficient curve of four different Reynolds number had consistent movements and shapes with the experimental curve. The lift coefficient curves of four Reynolds number were much closer with the experimental data, while drag coefficient curves differed slightly with the experimental data. This may be caused by the roughness of front edge or other factors. The flow separation didn’t occur around the airfoils of different attack angle, at this time Reynolds number was relatively small. The airfoil had a large velocity gradient around the front surface when attack angle was relatively large. The upper and lower velocity gradient of airfoil’s front edge differs with attack angle, pointing down or up. When attack angle is zero, the flow separates in the front edge of the airfoil and then combines in a certain distance of the rear airfoil.

REFERENCE