IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 14, Issue 5 Ver. I (Sep. - Oct. 2017), PP 80-85
www.iosrjournals.org

CFD Analysis of GOE 387Airfoil

*Mohamed A. Fouad Kandil[1], Abdelrady Okasha Elnady[2]
[1]*Teaching Assistant, Mechatronics Department, Faculty of Engineering, October 6 University*
[2]*Mechatronics Department, Faculty of Engineering /O6University, Egypt (Lecturer)*
Corresponding Author: Mohamed A. Fouad Kandil[1]

Abstract: *In this paper, we have obtained lift and drag forces for GOE 387 airfoil using CFD. The analysis of the two-dimensional subsonic flow over a GOE 387 airfoil at various angles of attack and operating at a Reynolds number of 3 x10^5 is presented. The geometry of the airfoil is created using ANSYS Design Modeler. CFD analysis is carried out using FLUENT 17.2 at various angles of attack from -5° to 20°. The motivation behind this research is to study the flow field over GOE 387 airfoil and obtain the aerodynamic characteristics of this airfoil. Lift coefficient and drag coefficient are plotted against theangle of attack. Variations of velocity distribution are plotted in form of contours for 3 x10^5 Reynolds number.*

Keywords*: Airfoil, Angle of Attack, CFD, Drag Coefficient, Lift Coefficient*

Date of Submission: 30-08-2017 Date of acceptance: 13-09-2017

I. Introduction

In the early 1800s, Sir George Cayleydiscovered that a curved surfaceproduced more lift than a similarsizedflat plate.What aviation pioneers discovered,which still holds true today,is that the most efficient way to dothis is to use a curved, streamlinedshape—the airfoil.Airfoil is defined as the cross-section of a body that is placed in an airstream in order to generate useful aerodynamic force.

The Wrightbrothers, many of the earlydesigners used “eyeball engineering”in developing or copying the airfoilshape used on their airplanes. Fromabout 1912, airfoil development andresearch moved to the wind tunnellaboratories found at the Universityof Göttingen in Germany,the Royal Aircraft Factory in theUnited Kingdom, and the NationalAdvisory Committee for Aeronautics(NACA) in the United States. Since then, the wind tunnel, complex mathematics, and the computer have all played important roles in airfoil development.

For the last threedecades, Computational Fluid Dynamics (CFD)has fascinated researchers, as it is a fast, accurate and reliablemethod of analyzing the variation of hydrodynamic propertiesof the flow over and within a body.Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve complex problems involving fluid flow.

Some of the recent interesting work in the study of airfoils has been discussed below.Patel, Karna S., et al [1]have obtained the drag and lift forces using CFD.Kevadiya, Mayurkumar[2] investigated NACA 4412 airfoil at various angles of attack from 0° to 12° using CFD analysis. Variations of pressure coefficient are plotted in form of contour for1 ×105 Reynolds number.

Patil et al. [3] investigated the effect of low Reynolds number on lift and drag for the wind turbine blade. They found out as Reynolds number increases, lift and drag forces increases.Haque et al. [4] conducted various experimental studies to understand the effects of Reynolds number and angle of attack in flow analysis. Yao et al. [5] studied the aerodynamic performance of wind turbine airfoils and compared the numerical results with experimental data. The effect of transonic flow over an airfoil has been studied and a comparative analysis hasbeendone to analyze the variation of theangle of attack and Mach number [6].

In the light of the review of the existing literature, the present study aims to analyze the flow field for GOE 387 airfoil at various angles of attack with constant Reynolds number of 3 x10^5. The flow was obtained by solving the steady-state governing equations of continuity and momentum conservation with Transition k-kl-omega turbulence model.

Lift Coefficient (CL): It is a dimensionless quantity that relates the lift generated by airfoil to the fluid density around the body, the fluid velocity, and an associated reference area.

$$C_L = \frac{L}{\frac{1}{2}\rho v^2 A}$$

Where: L is the lift force,ρ is the density of air, V is inlet velocity of air, A is the area of theairfoil.

Drag Coefficient (CD): It is a dimensional quantity that is used to quantify the drag or resistance of an object in a fluid environment.

$$C_D = \frac{D}{\frac{1}{2}\rho v^2 A}$$

Where: D is the drag force, ρ is the density of air, V is inlet velocity of air, A is the area of theairfoil.

II. Geometry And Mesh Generation

The geometry of GOE 387 is shown in Figure 1. For discretization of the computational domain, an unstructured mesh with thebody of influence centered on the airfoil and rectangular path were selected. The mesh used for the analysis is shown in Figures 2 and 3, Pressure based steady state solver with Transition k-kl-omega turbulence model is used for analysis.

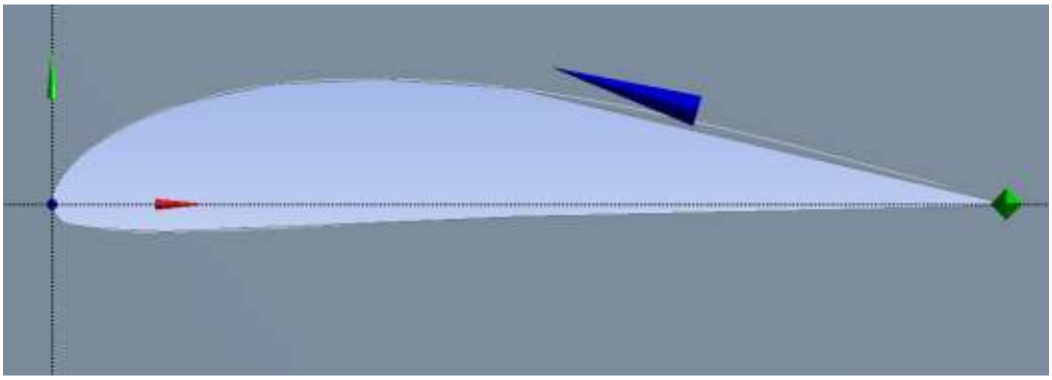

Figure 1: Geometry of GOE 387Airfoil

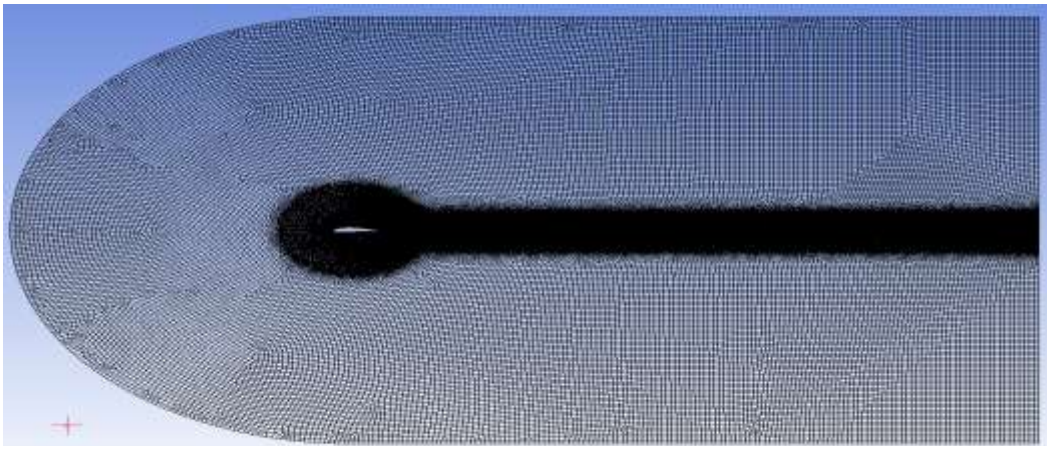

Figure 2: Completed Mesh

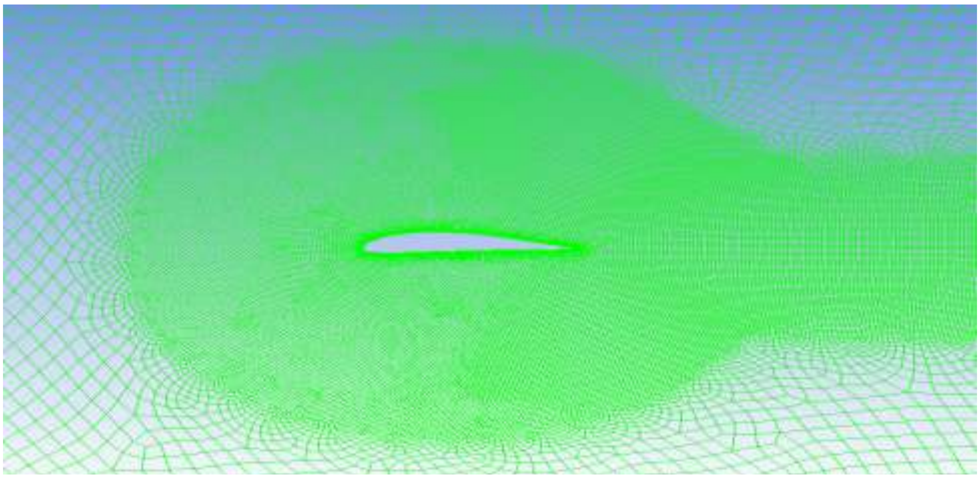

Figure 3: Mesh of the computational domain

Model Data:	Number of Nodes	97961
	Number of Elements	97396

III. Inputs And Boundary Conditions

The problem consists of flow around an airfoil at various angles of attack (-5, 0, 5, 10, 15, 20 degrees). The inputs and boundary conditions are shown in Table1.

Input	Value
Solver	Pressure based
State	Steady
Vicious model	Transition k-kl-omega
Material	Air
Density	1.225
Viscosity	1.7894e-05
Reynold Number	3×10^5
Inlet velocity	4.3822 m/s
Chord-length	1 m
Pressure velocity coupling	Coupled

Table 1: Inputs and boundary conditions

IV. Results And Discussion

4.1 Contours of velocity magnitude

The contours of velocity magnitude obtained for various angles of attack from CFD simulations are shown in the following Figures 4,5,6,7,8, and 9. On the leading edge, we can see the stagnation point where the velocity of flow is nearly zero. The fluid accelerates on the upper surface of the airfoil while the velocity of the fluid decreases along the lower surface of the airfoil.

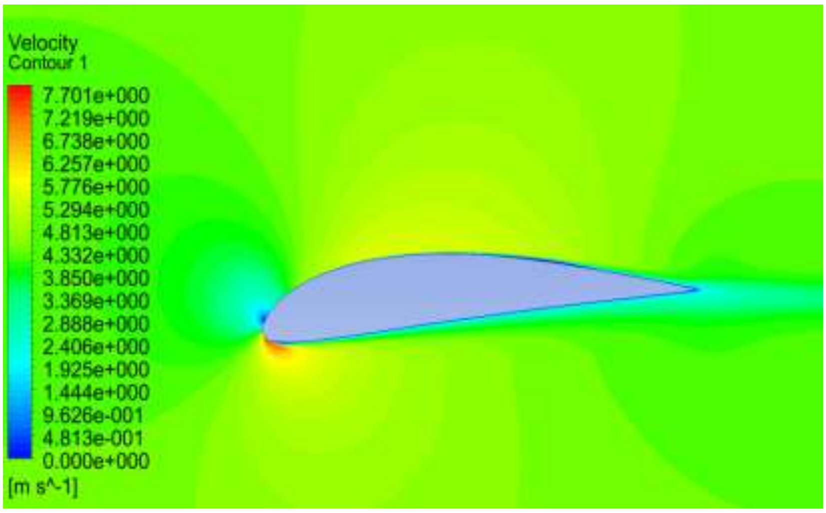

Figure 4: Contours of velocity magnitude at -5 degrees of angle of attack

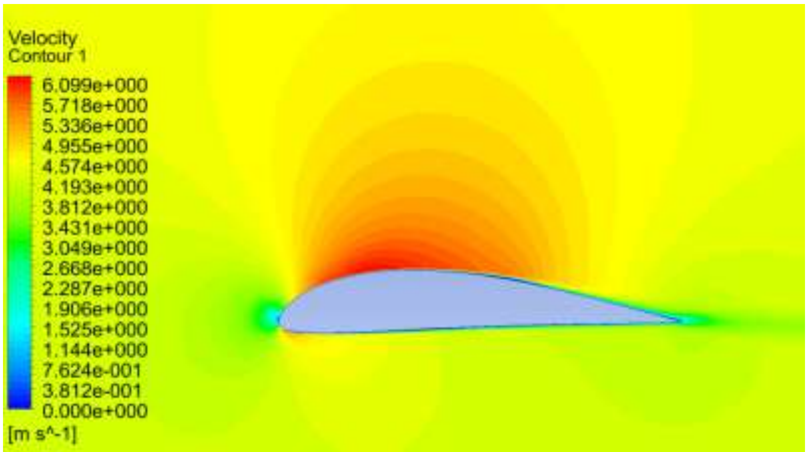

Figure 5: Contours of velocity magnitude at 0 degrees of angle of attack

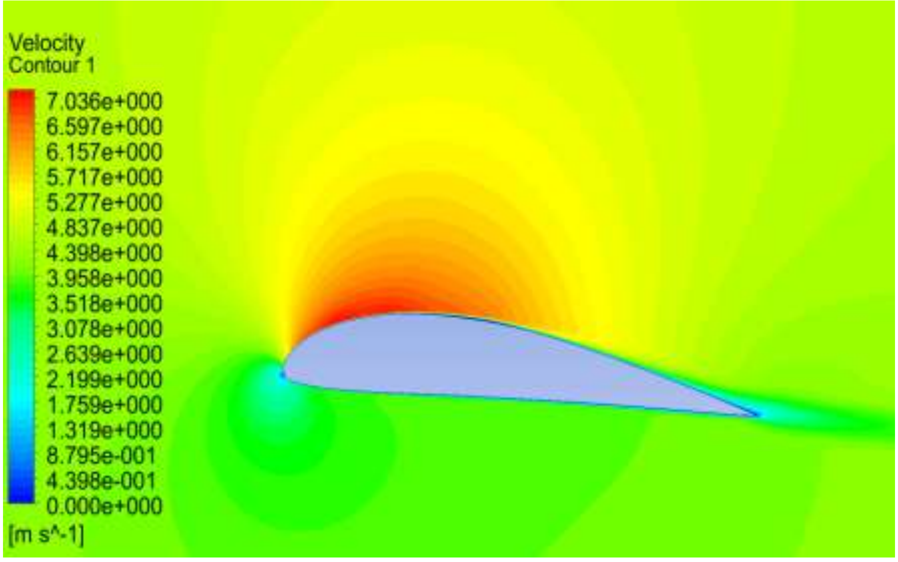

Figure 6: Contours of velocity magnitude at 5 degrees of angle of attack

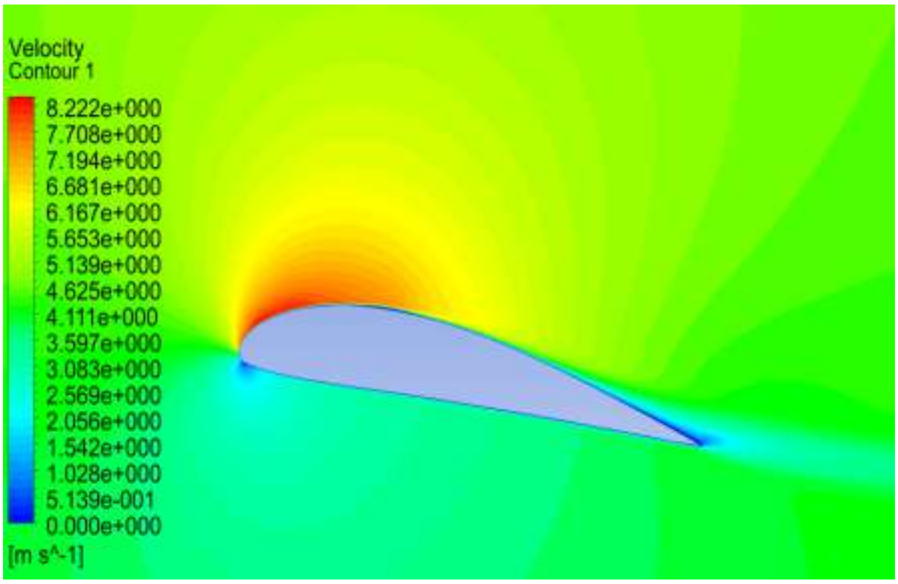

Figure 7: Contours of velocity magnitude at 10 degrees of angle of attack

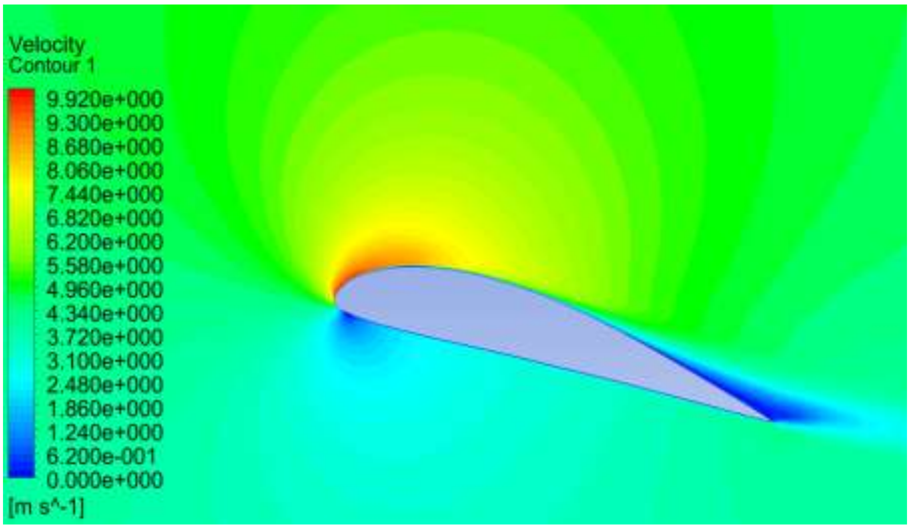

Figure 8: Contours of velocity magnitude at 15 degrees of angle of attack

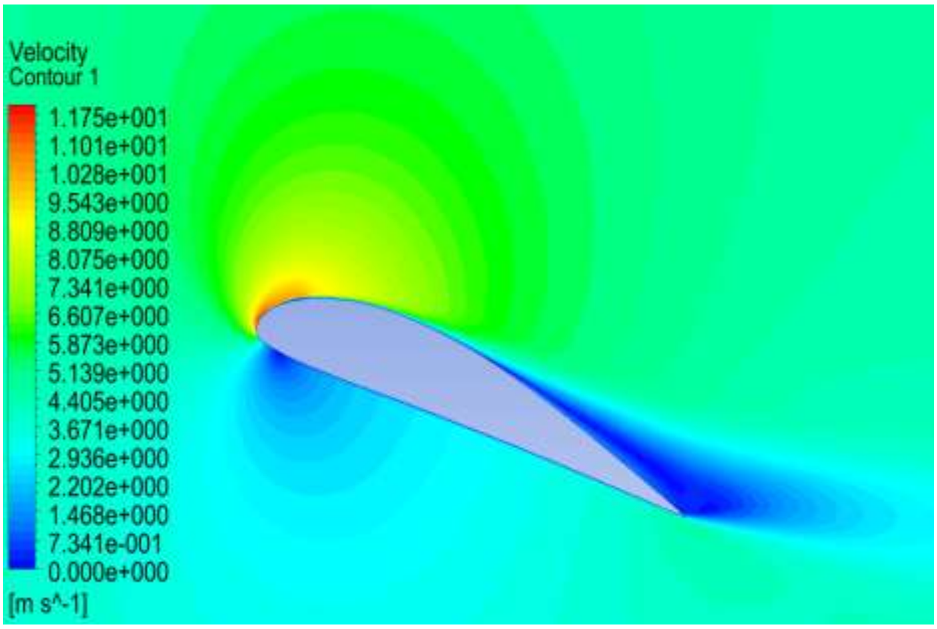

Figure 9: Contours of velocity magnitude at 20 degrees of angle of attack

4.2 Curve of lift and drag coefficient

Lift coefficient, drag coefficient, and C_L/C_D ratio at various angles of attack are presented in Table 2

Lift Coefficients Cl	Drag Coefficients Cd	Cl/Cd	Angle of Attack α
2.4476e-02	2.0620e-02	1.18700291	**-5**
5.1434e-01	1.1540e-02	44.57019064	**0**
1.1023e+00	1.5786e-02	69.82769543	**5**
1.5765e+00	2.5907e-02	60.85227931	**10**
1.9256e+00	4.5407e-02	42.40755831	**15**
1.7738e+00	1.0103e-01	17.55716124	**20**

Table 2: Lift, Drag, and Lift to Drag Coefficients

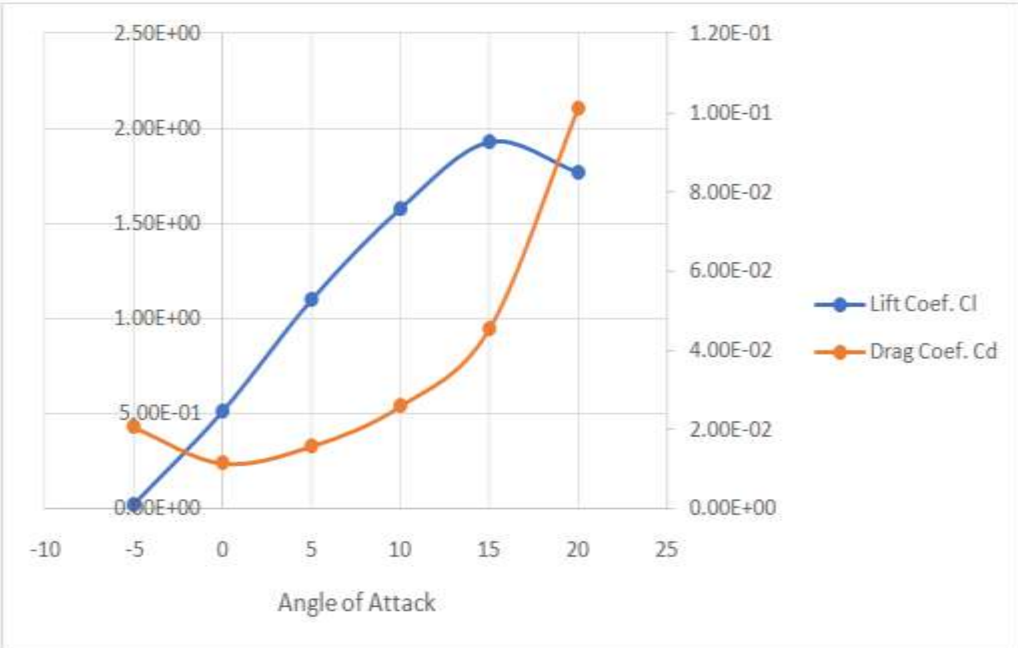

Figure 10: Curve of Lift & Drag Coefficients vs. Angle of Attack

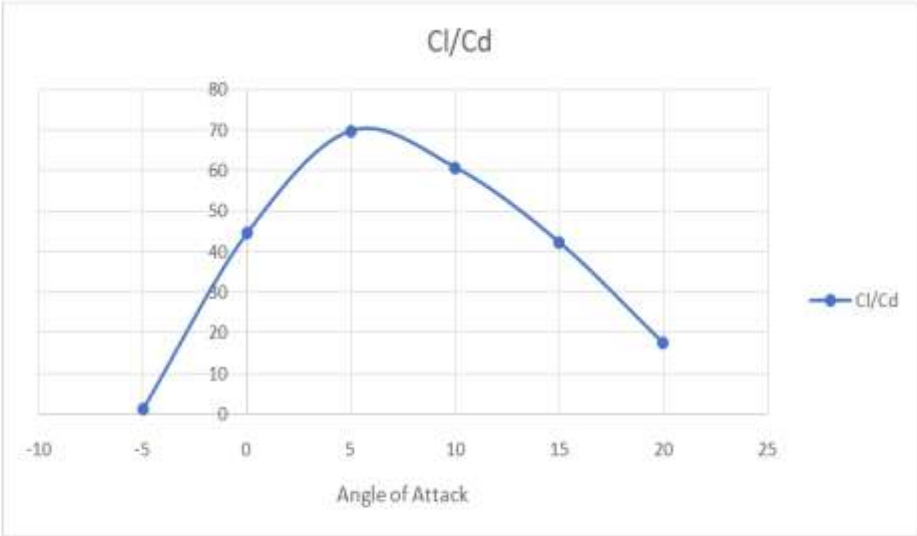

Figure 11: Curve of Lift to Drag Coefficients vs. Angle of Attack

The GOE 387 airfoil characteristics are summarized in Table 3.

C_{Lmax}	1.93
C_{Lopt}	1.1
(C_L/C_D)	69.83
Maximum $C_{Langle}\alpha$(stall)	15
Zero-Lift angle	-5

Table 3:GOE airfoil Characteristics

V. Conclusion

With the help of CFD software Ansys-Fluent, successful analysis of theaerodynamic performance of GOE 387 airfoil has been carried at various angles of attack (-5, 0, 5, 10, 15, 20 degrees) with constant Reynolds number (3 x10^5) using the Transition k-kl-omega turbulence model. It is seen that the velocity of theupper surface is higher than the velocity of the lower surface. It is observed that to increase the value of lift force and lift coefficient we have to increase the value of theangle of attack. This leads to rise in drag force and drag coefficient as well, but the increase in drag force and drag coefficient is quite low in comparison to lift force and lift coefficient.

References

[1]. Patel, Karna S., et al. "CFD Analysis of an Aerofoil." International Journal of Engineering Research 3.3 (2014): 154-158.
[2]. Kevadiya, Mayurkumar. "CFD Analysis of Pressure Coefficient for NACA 4412." International Journal of Engineering Trends and Technology, Chennai 4.5 (2013): 2041-2043.
[3]. Bhushan S. Patil, Hitesh R. Thakare, Computational Fluid Dynamics Analysis of Wind Turbine Blade at Various Angles of Attack and different Reynolds Number, Procedia Engineering, 127(2015), 1363-1369
[4]. Nazmul Haque, Mohammad Ali, Ismat Ara, Experimental Investigation on the performance of NACA 4412 aerofoil with curved leading edge plan form, Procedia Engineering, 105(2015), 232-240
[5]. Jin Yao, Weibin Yuan, Jianliang Wang, Jianbin Xie, Haifeng Zhou, MingjunPeng, young Sun, Numerical simulation of aerodynamic performance for two-dimensional wind turbine airfoils, Procedia Engineering, 31(2012), 80-86
[6]. Novel Kumar Sahu, Mr. Shadab Imam, Analysis of Transonic Flow over an Airfoil NACA0012 using CFD, International Journal of Innovative Science, Engineering & Technology, Vol. 2 Issue 4, April 2015

Mohamed A. Fouad Kandil. “CFD Analysis of GOE 387Airfoil.” IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) , vol. 14, no. 5, 2017, pp. 80–85.