ZIBELINE INTERNATIONAL PUBLISHING

Journal of Mechanical Engineering Research & Developments (JMERD)

ISSN: 1024-1752 (Print)

CODEN: JERDFO

DOI : http://doi.org/10.26480/jmerd.01.2019.87.92

RESEARCH ARTICLE

COMPUTATIONAL STUDY OF FLOW CHARACTERISTICS OVER HIGH LIFT AIRFOIL AT VARIOUS ANGLES OF ATTACK

Ali H. Mutaib*[1], Amjed AL-Khateeb[2], Mohammed K. Khashan[2], Furkan Kamil[1]

[1]Department of Aeronautical techniques, Engineering Technical College of Al-Najaf, Al-Furat Al-Awsat Technical University, Najaf, 31001, Iraq.
[2]Department of Aeronautical techniques, Najaf technical institute, Al-Furat Al-Awsat Technical University, Najaf, 31001, Iraq.
**Corresponding Author Email: coj.aliha@atu.edu.iq*

ARTICLE DETAILS

Article History:

Received 04 January 2019
Accepted 08 February 2019
Available online 13 February 2019

ABSTRACT

A numerical simulation aiming to study and investigate flow behavior over a high lift low speed airfoil type AG-16 at various angles of attack ranging from 0 degree to 15 degree and constant Reynolds number "39,709" was performed. Computational fluid dynamic CFD technique was applied to study the flow characteristics around the complicated shape airfoil; for better calculation results and smoother meshing generation process unstructured mesh with tetrahedron grid element was used. SOLIDWORKS program was used for modeling requirements; the modeling process includes designing three-dimensional wing with airfoil cross section AG-16 to take into account the influence of third dimension on the flow variables. The magnitude of adverse pressure gradient and flow separation point was investigated, the critical/stall angle of attack of this type airfoil was specified and the aerodynamic efficiency was studied and analyzed with respect to various angles of attack. The numerical results illustrate that the lift coefficient rise rapidly with decreasing angle of attack. The critical angle of attack that gives the maximum lift was recorded at 14 degree on the other hand the drag increased rapidly at high angles of attack due to flow separation, the optimum angle of attack that provide the maximum aerodynamic ratio was 4 degree.

KEYWORDS

Airfoil, Aerodynamic Performance, Angle of Attack, Numerical Analysis, Computational Study

1. INTRODUCTION

As air flows around a body, it effects on this body by the action of aerodynamic forces, lift and drag are the most significant forces that affect directly on the body. The lift is produced according to the pressure difference between the upper and lower surfaces, and its direction is perpendicular to the flow motion, the drag is produced due to interaction of flow with body surface, and its direction is opposite to the flow motion. An attempt was carried out by a researcher to investigation of flow around a set of airfoils with different shapes and thickness by applying Reynolds-Averaged Navier-Stokes equations (RANS) equations at high Reynolds number and low turbulence intensity, the results shown there is no significant effect of compressibility on the lift coefficient even at high angles of attack and also does not influence on the maximum lift coefficient [1]. The flow behavior over NACA 0012 airfoil was studied and analyzed numerically by a researcher, k-ω shear stress transport (SST) model was applied with turbulence intensities 1% and 5% to predict the characteristics of flow accurately, according to the results they concluded, when Mach number increased the lift coefficient increases but drag coefficient remains constant, and also when the velocity approaching to the sonic limit the lift coefficient decreased rapidly with sudden increase in drag coefficient [2].

An experimental study supported by computational approach using CosmosFlo Works software was carried out by a previous researcher, the pressure distribution on the upper and lower surface of airfoil type Clark-Y at various angles of attack in low speed wind tunnel was investigated, labView software was used for measurement the distribution of pressure by controlling the Scanivalve solenoid, the results showed when angle of attack increased from 0 to 15 degrees the lift coefficient increased, and then it decreased rapidly when the angle of attack increased due to separation of flow from the upper surface of airfoil [3]. A recent scholars presents a numerical simulation goals to analyzing the subsonic flow with constant Reynolds number "3×106 over NACA0012 airfoil at different angle of attack, the analysis method includes solving the governing equations of continuity and momentum combined with one of three turbulence models "Spalart-Allmaras, Realizable k -ε and k -ω shear stress transport (SST)",their study focused on two areas that, transition point prediction and turbulence modeling [4]. The results found that when the angle of attack is high the turbulence models of the commercial CFD doesn't give accurate results.

An experimental study carried by a researcher's goals to study the flow characteristics around NACA 4415 airfoil at constant Reynolds number and different angle of attack take into consideration the effect of ground clearance of the trailing edge, the results found there is a strong suction effect on the lower surface when the angle of attack ranging from 0 to 2.5 degree at small ground clearance, laminar separation observed at the lower surface wall ahead of the trailing edge, when the angle of attack rise to 10 degree the magnitude of the adverse pressure on the upper surface will be increased rapidly causes separation of flow from the upper surface, which in turn the lift will decrease and drag will increase [5]. A numerical attempt applied by a researcher, the numerical study includes applying conformal mapping technique for modeling the flowing fluid around three types of NACA airfoil 0012, 2215 and 4412, another researcher's transformation was used to link the flow solution of cylindrical to that of an airfoil, the numerical calculations were compared with lift calculations generated from thin airfoil method, the results shown that lift data for NACA 0012 airfoil fit with the expected results, while NACA airfoils 2215 and 4412 suffer disagreement in lift calculation at low angles of attack [6].

Unsteady numerical investigation performed by a researcher's aims to study the aerodynamic behavior of four digit NACA airfoil at different thickness "8%, 12% and 16%" with ultra-low Reynolds number of 1000, the flow characteristics at various angles of attack was investigated, the numerical calculations found that the magnitude of lift to drag ratio generated by thinner airfoil was greater than thick one, and thinner airfoil generate high lift before reach to stall condition [7].

A numerical simulation aimed to study and analyzing the aerodynamic characteristics of NACA 2415 airfoil was carried out by a scholar, the study focused on the designing a wing with better aerodynamic performance [8]. Selecting an appropriate mesh type is the most significant parameter that effect strongly on the accuracy of numerical results, a researcher applied unstructured mesh with tetrahedron elements for meshing the complicated geometry of hydro-propeller [9].

The present work deals with study and investigation of flow behavior over high lift low speed airfoil type Ag-16 and analyzing the aerodynamic characteristics of this airfoil at various angles of attack and constant Reynolds number.

2. AERODYNAMIC CHARACTERISTICS OF AIRFOIL

When air strike an airfoil the flow layers will splits to surround the airfoil, two forces will produce one works to lifting the body and effect vertically on the flow direction; known as lift force, and the other makes to obstruct the flow motion and effect horizontally with a direction opposite to the flow motion; known as drag force. the magnitude of lift force depends on the geometric configuration of the airfoil especially " camber ness " and the angle of attack, while drag force effected by the surface roughness, angle of attack, flow speed and geometric shape of an airfoil. The mathematical relations of lift and drag forces illustrate in the following equations.

$$L = \frac{1}{2}\rho V^2 S C_L \quad (1)$$

$$D = \frac{1}{2}\rho V^2 S C_D \quad (2)$$

3. THE NUMERICAL SIMULATION

Computational fluid technique was adopted in this study represented by ANSYS-FLUENT 16.1, the numerical simulation includes three main steps; modeling the geometry, meshing the computational domain and numerical calculation.

3.1 Modeling of Airfoil Geometry

Airfoil geometry was created using SOLIDWORK program; the modeling process includes two domains; the air duct or wind tunnel and three-dimensional wing with airfoil type AG-16, as shown in Figure 1, the purpose of modeling three dimensional wing is to take into account the effect of third axis on the flow properties. Six cases were built with respect to angle of attack as shown in Figure 2. Figure 3 shows the dimensions of wind tunnel and table 1 illustrates the geometric dimensions of wing, while table 2 illustrates flow properties of air.

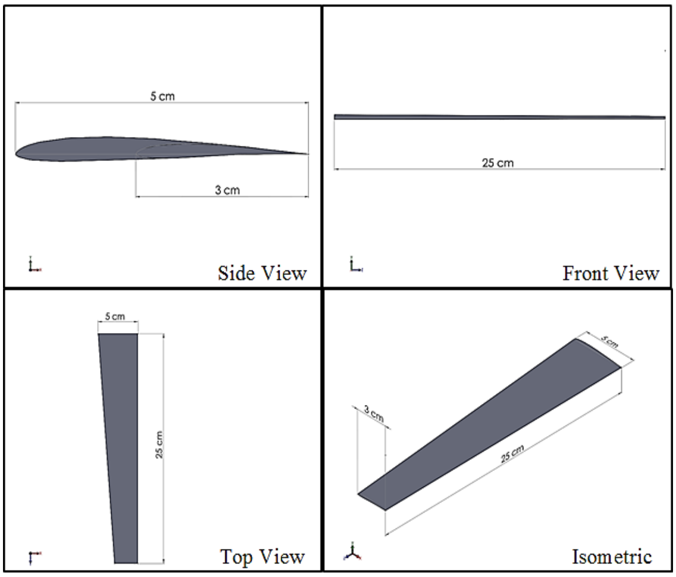

Figure 1: Modeling of three-dimensional wing

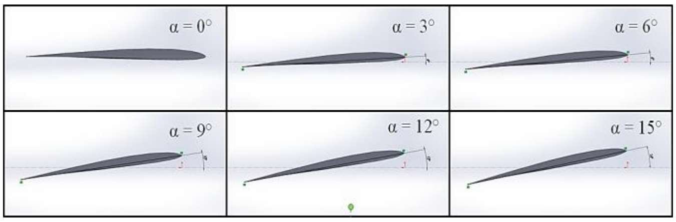

Figure 2: Modeling of wing with respect to angles of attack

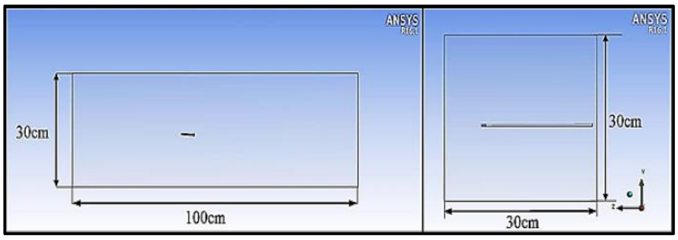

Figure 3: Modeling of wind tunnel (Duct)

Table 1: Geometrical Specifications of Wing

NO.	GEOMETRICAL SPECIFICATION	MAGNITUDE	UNIT
1	Airfoil chord at wing root (cr)	5	cm
2	Airfoil chord at wing tip (ct)	3	cm
3	Airfoil maximum thickness (t)	0.35	cm
4	Wing span (b)	50	cm
5	Wing taper ratio (λ)	0.6	-
6	Aspect ratio (AR)	12.5	-

Table 2: Physical Properties of Air

NO.	PHYSICAL PROPERTY	MAGNITUDE	UNIT
1	Stream velocity (V)	15	m/s2
2	Air density (ρ)	1.225	Kg/m3
3	Air viscosity (μ)	18.13×10-6	Pa.s
4	Air temperature (T)	20	°C
5	Reynolds number	39,709	

3.2 Meshing the Computational Domain

Unstructured mesh with tetrahedron element was adopted in this study, because of it offers better numerical results and running the calculation process smoothly. Refining mesh technique was applied at the leading edge, trailing edge, wing tip and wing root due to these locations are suffers high pressure gradient, as shown in figure 4. Mesh dependency technique was carried out to check the influence of elements number on the numerical results accuracy, lift coefficient was adopted in this work as mesh dependency criterion to check the results accuracy for three types of mesh differs in the number of elements “coarse, medium and fine” mesh as illustrated in table 3. The percentage of error for the last two cases was 3.79%, it is too small as compared with number of elements, so the medium type of mesh was selected and applied for all cases due to its decrease the calculations time and give acceptable result accuracy as compared with other grid types.

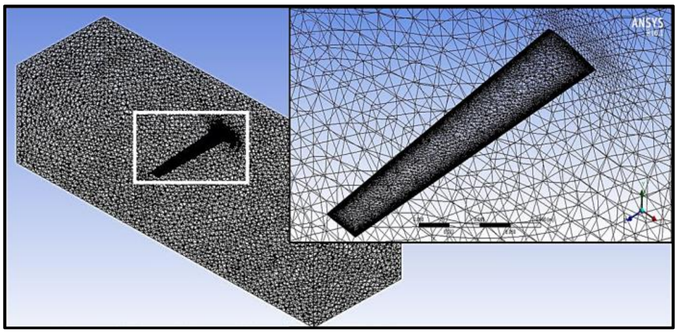

Figure 4: Meshing of computational domain

Table 3: Mesh Dependencies

No.	Mesh Type	Number Of Elements	Coefficient Of Lift
1	Coarse	983,943	0.00773
2	medium	4,280,225	0.00685
3	fine	5,642,315	0.00659

3.3 Boundary Conditions

The purpose of specifying boundary conditions is to introduce the flow variables on computational domain boundaries. In this work three boundary conditions were adopted; velocity inlet was used to specify the inlet section of wind tunnel, pressure outlet applied on the outlet section of wind tunnel, while wind tunnel sides and wing sides specified as wall

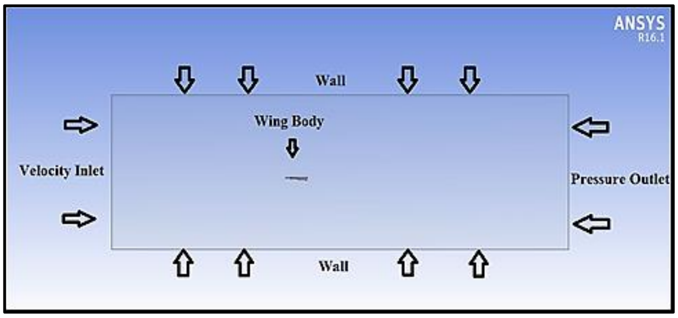

Figure 5: Specifying boundary condition

3.4 Turbulence Modeling

The effect of turbulence of airflow over three-dimensional wing is very important and should be taken into account during airflow analyzing process. To predict the development of turbulence in this study k-ε model was applied as turbulence model.

3.5 Numerical Calculations

Computational fluid dynamic technique based on finite volume method was applied in this study represented by ANSYS-FLUENT 16.1, the flow over three dimensional wing with airfoil type AG-16 was analyzed and simulated by converting the physical model to computational domain throughout meshing it to finite number of elements. The governing equations for continuity, momentum and turbulence were converted to algebraic form. The equations residuals were specified and selected to equal 10-3 that represented the end of iteration, 2400 iterations were adopted in this study to get accurate results, an additional 500 iterations are desired in case of the residuals not minimize the selected values.

4. RESULTS AND DISCUSSION

A numerical analysis of aerodynamic performance for AG-16 airfoil at different angles of attack ranging from 0 degree to 15 degree was performed, the numerical solution was obtained using ANSYS-FLUENT program. The numerical results of pressure, turbulence intensity and aerodynamic characteristics such as lift and drag were inserted in the form of Graphs and Figures.

4.1 Pressure Distribution around Airfoil

High lift airfoils characterized by having pronounced camber, the static pressure at the upper surface will be lower than its magnitude at the lower surface and this will create a pressure difference between both surfaces. Figures 6 shows cross section contours of pressure and illustrates the distribution of pressure around airfoil with different angles of attack ranging from 0 degree to 15 degree. Figures 6 -11 indicate at constant Reynolds number, when angle of attack raises the pressure difference will be augmented because of increasing projected area of the wing

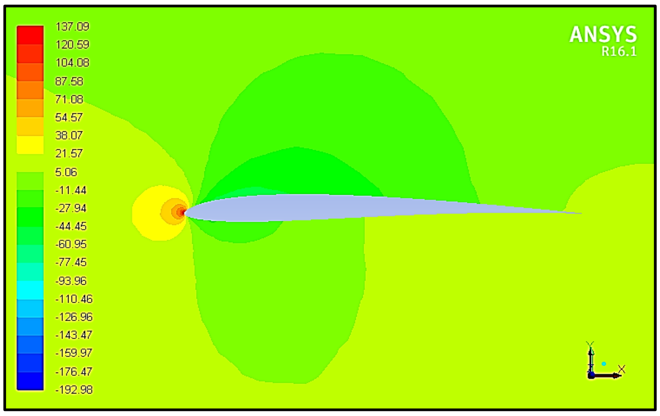

Figure 6: Pressure distribution contour at 0 degree angle of attack

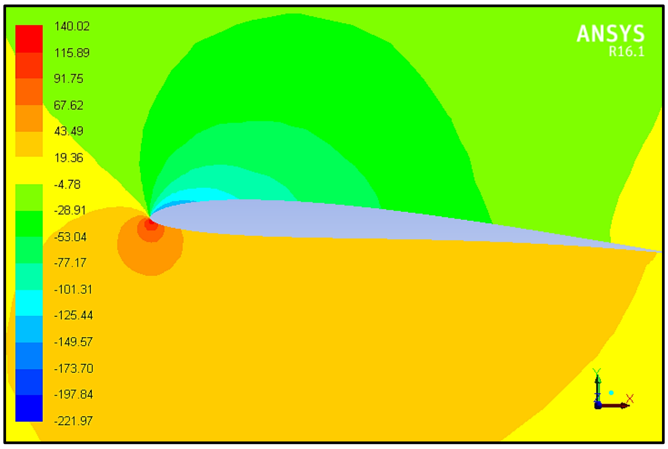

Figure 7: Pressure Distribution Contour at 3 Degree Angle of Attack

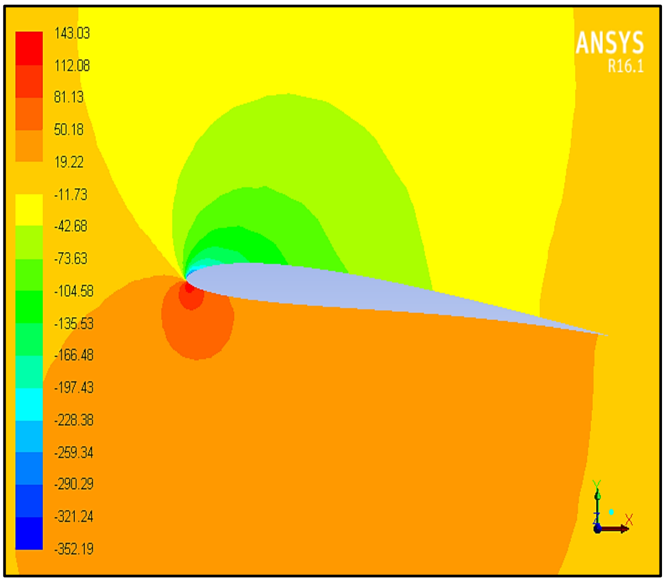

Figure 8: Pressure Distribution Contour at 6 Degree Angle of Attack

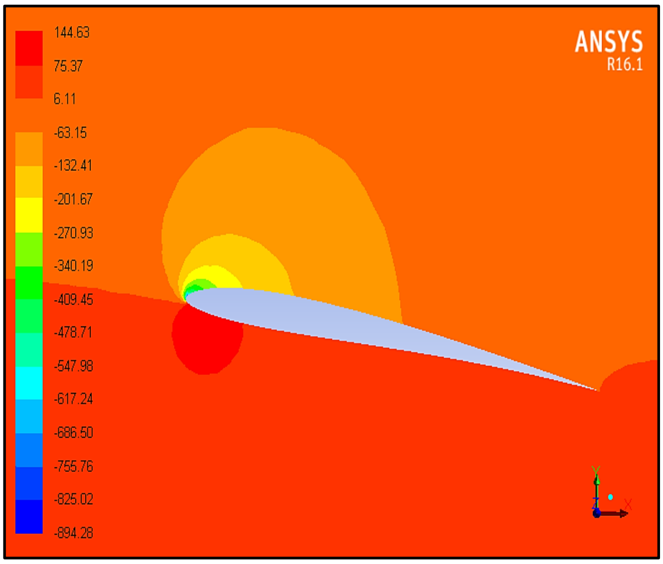

Figure 9: Pressure Distribution Contour at 9 Degree Angle of Attack

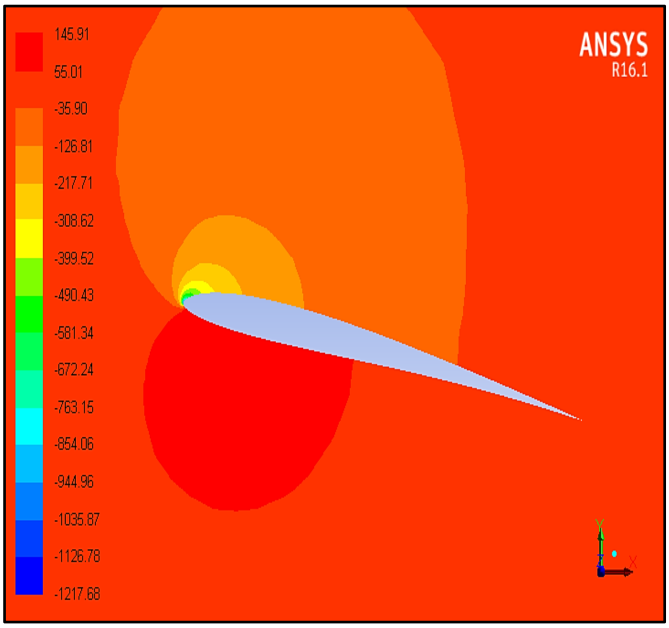

Figure 10: Pressure Distribution Contour at 12 Degree Angle of Attack

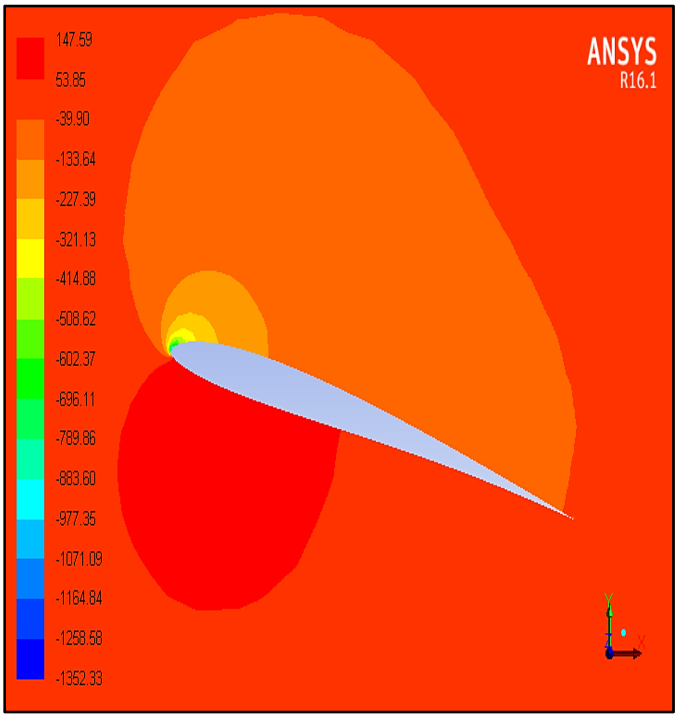

Figure 11: Pressure Distribution Contour at 15 Degree Angle of Attack

4.2 Velocity Distribution around Airfoil

Figures 12 to 17 show cross section contours of velocity and indicates that the velocity above the upper surface is greater than the lower surface due to airfoil camber ness, and when the angle of attack raises the flow will separate from the upper surface and the separation intensity will be grow up as the angle of attack increases reaching a critical limit. The separation point moves forward towards airfoil leading edge as the angle of attack increase and this will make the high pressure which exists in the lower surface move up to the upper surface.

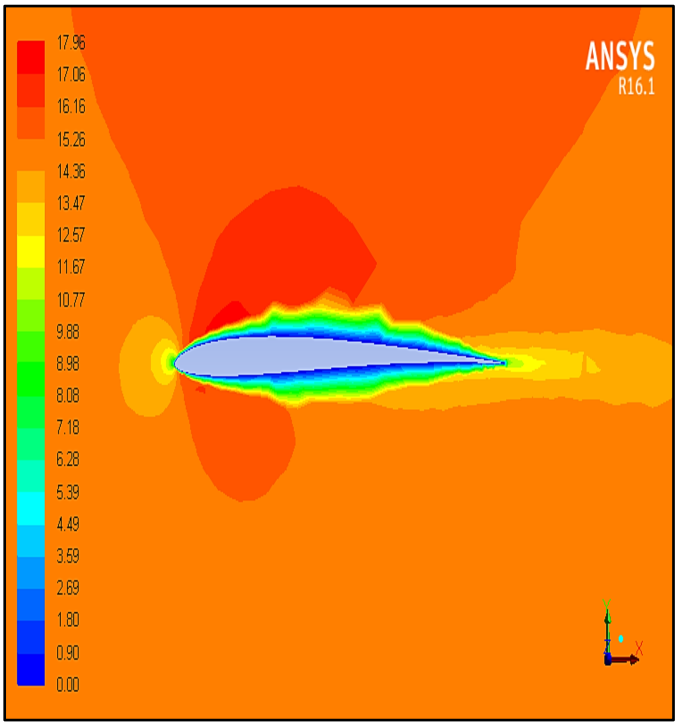

Figure 12: Velocity Distribution Contour at 0 Degree Angle of Attack

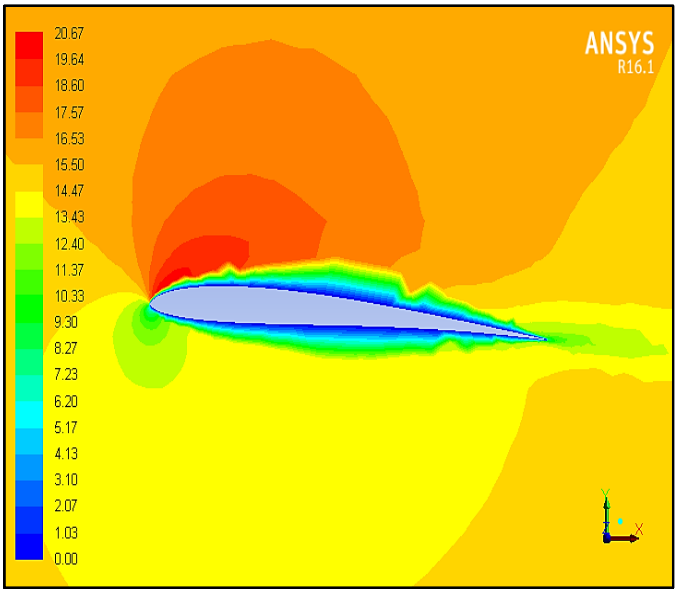

Figure 13: Velocity Distribution Contour at 3 Degree Angle of Attack

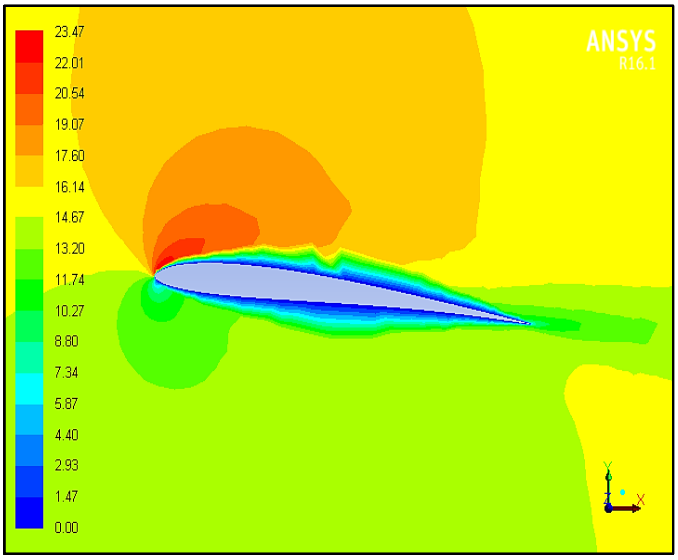

Figure 14: Velocity Distribution Contour at 6 Degree Angle of Attack

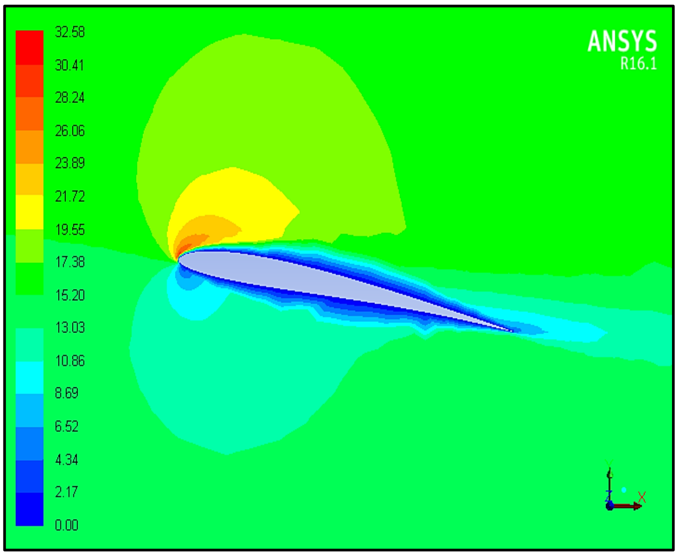

Figure 15: Velocity Distribution Contour at 9 Degree Angle of Attack

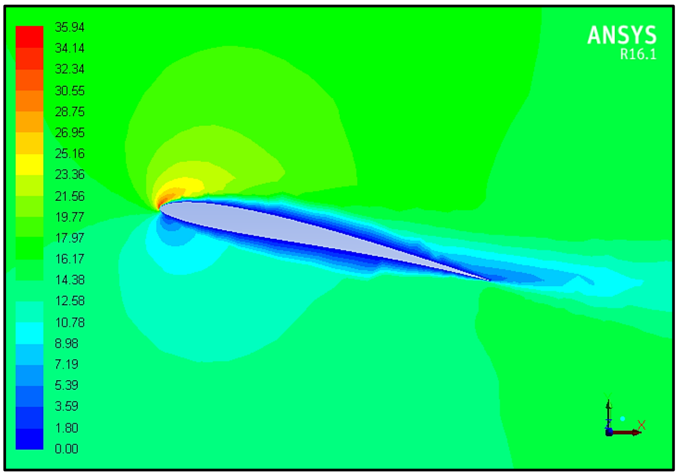

Figure 16: Velocity Distribution Contour at 12 Degree Angle of Attack

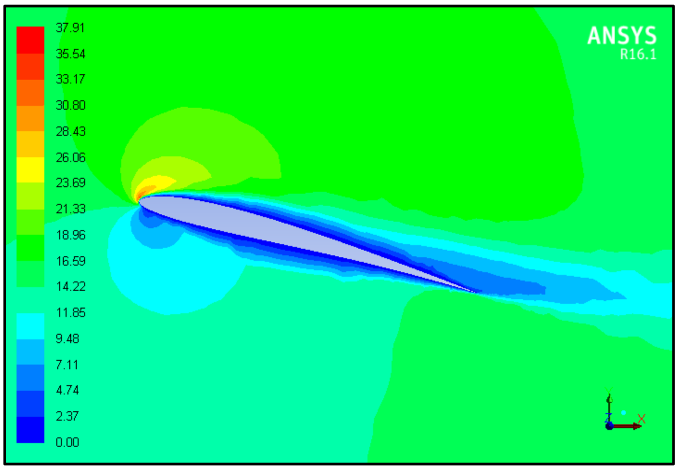

Figure 17: Velocity Distribution Contour at 15 Degree Angle of Attack

4.3 Lift Coefficient

The airfoil camber ness and angle of attack are the most effective parameters on the lift coefficient magnitude. Figure 18 presents a relation between lift coefficient adverse angle of attack and illustrates that with increasing angle of attack the lift coefficient increases rapidly reaching its maximum value of 0.0116 when the angle of attack equal to 14 degree, then drop with further raising of the angle of attack due to maximum moving of separation point towards the leading edge accompanied by large increase in pressure over the top surface causing abrupt decrease in the lift.

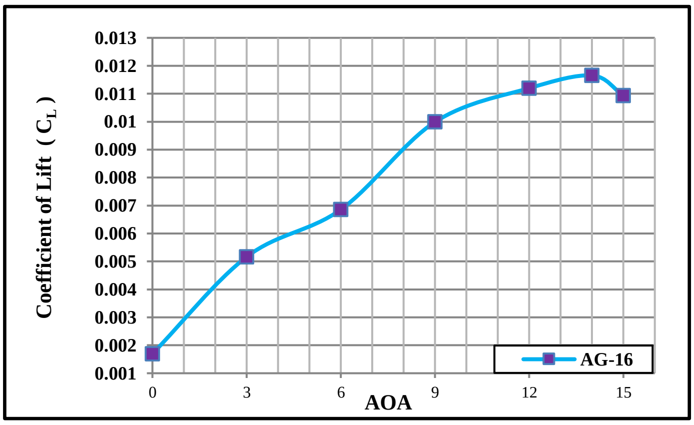

Figure 18: Lift Coefficient Adverse Angle of Attack

Cite The Article: Ali H. Mutaib, Amjed AL-Khateeb, Mohammed K. Khashan, Furkan Kamil (2019). Computational Study Of Flow Characteristics Over High Lift Airfoil At Various Angles Of Attack. *Journal of Mechanical Engineering Research & Developments*, 42(1): 87-92.

4.4 Drag Coefficient

Figure 19 shows the relation between the drag coefficient adverse angle of attack and indicates that at small angle of attack the drag coefficient does not change significantly, but when the angle of attack increase the separation point moves forward reaching to the leading edge. The value of drag coefficient will rise rapidly due to turbulence effect resulting from the separation of flow at the upper surface of the wing.

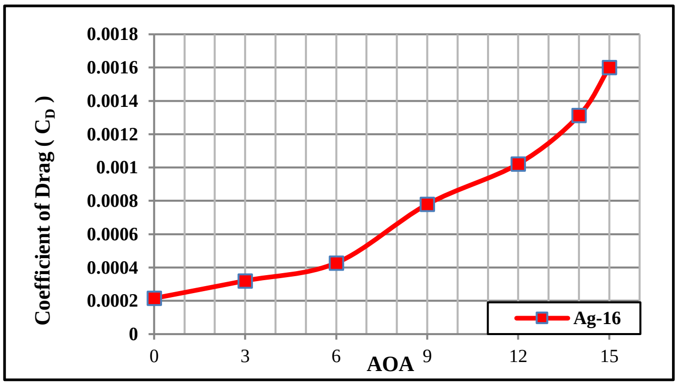

Figure 19: Drag Coefficient Adverse Angle of Attack

4.5 Aerodynamic Efficiency

The airfoil is characterized by its ability to produce a lift force which is greater than the drag force. Figure 10 presents the relation between aerodynamic efficiency (lift/drag ratio) and adverse angle of attack and shows that the optimum angle of attack which produces the highest aerodynamic ratio is equal to 4 degree.

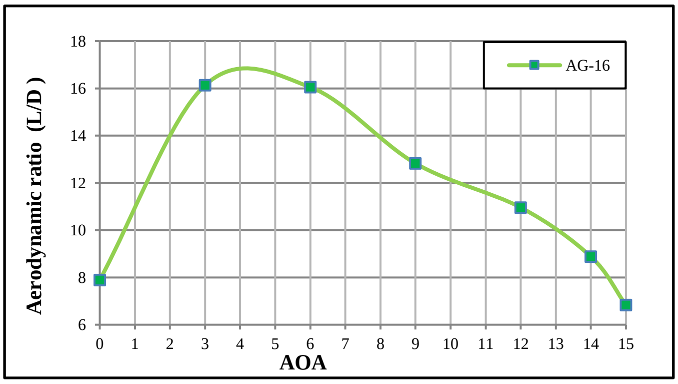

Figure 20: Aerodynamic Efficiency of AG-16 Airfoil

5. CONCLUSIONS

A numerical simulation based on computational fluid dynamic represented by ANSYS-FLUENT 16.1 that used finite volume technique aims to study and investigate the aerodynamic characteristics of high lift airfoil type AG-16 with different angles of attack ranging from 0 degree to 15 degree at constant Reynolds number, the numerical results show the following points: -

1. The lift coefficient increases rapidly as the angle of attack increases reaching a critical limit then suddenly decreases with continues increasing of the angle of attack due to separation of flow at the upper surface.
2. The maximum lift coefficient for high lift airfoil type AG-16 recorded at 0.0116.
3. The stall angle of attack or critical angle of attack for AG-16 airfoil recorded is equal to 14 degree.
4. The drag coefficient increased slightly with small angles of attack and then rapidly when the angle of attack reaches the critical limit due to high turbulence resulting from separation of flow at the upper surface.
5. The drag coefficient magnitude is equal to 0.0013 when the angle of attack reaches the stall limit.
6. The optimum angle of attack that gives the maximum aerodynamic ratio was 4 degree.

REFERENCES

[1] Aleksey, A.M., Eugeniy, V.K., Andrey, V.G. 2017. Calculations of flow around airfoils using two-dimensional RANS: an analysis of the reduction in accuracy, St. Petersburg Polytechnical University Journal: Physics and Mathematics, 3, 15-21.

[2] Ahmed, T., Amin, M.T., Rafiul Islam, S.M., Ahmed, S. 2013. Computational Study of Flow Around a NACA 0012 Wing Flapped at Different Flap Angles with Varying Mach Numbers, Global Journal of Researches in Engineering, General Engineering, 13.

[3] Matsson, J. 2007. A Student Project on Airfoil Performance, American Society for Engineering Education.

[4] Douvi, C.E., Tsavalos, I. 2012. Athanasios and Margaris P. Dionissios, Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil, 4, 100-111.

[5] Ahmed, M.R. 2005. Aerodynamics of a Cambered Airfoil in Ground Effect, International Journal of Fluid Mechanics Research, 32, 157-183.

[6] Kapania, N.R., Terracciano, K., Taylor, S. 2008. Modeling the Fluid Flow around Airfoils Using Conformal Mapping, 70-99.

[7] Uddin, E., Naseem, M.A., Khalid, S.U., Mubashar, A., Shah, S.R. 2017. Investigation of the flow around uncambered airfoils at 1000 Reynolds number using computational fluid dynamics for micro air vehicles, the 2017 World congress on Advances in Structural Engineering and Mechanics, 28 August – 1 September 2017, IIsan (Seoul), Korea.

[8] Jain, R., Jain, S., Bajpai, L. 2016. Investigation on 3-D Wing of commercial Aeroplane with Aerofoil NACA 2415 Using CFD Fluent, 3, 243-249.

[9] Khashan, M.K., Al-Khateeb, A., Mutaib, A.H., Al-Zaini, E.O. 2018. Influence Of Increasing Number Of Blades On The Internal Structure Of Hydro Propeller, International Journal of Mechanical Engineering and Technology (IJMET), 9, 1434–1443.

Nomenclature

CL	Lift coefficient	
CD	Drag coefficient	
L	Lift force	N
D	Drag force	N
S	Wing area	m2
V	Velocity	m2/s

Greek letters

ρ	Density	kg/m3