

CFD Analysis of NACA 63-018 Airfoil at Different Reynolds-Number

Abdul Azeez VP^{#1}, John paul^{#2}

^{#1} M.Tech scholar, ^{#2} Asst. Professor

^{#1,#2} Department of Mechanical Engineering, MESCE, Kuttippuram, Kerala, India

Abstract— This paper numerically investigated the evolution of the aerodynamic performance parameters and flow field of two-dimensional airfoils called NACA 63-018 at different Reynolds numbers 3×10^6 , 6×10^6 and 9×10^6 with different angle of attack ranging from 0° to 16° . The airfoil was modeled by using CATIA V5 and the analysis was carried out by using CFD technique ANSYS FLUENT 14.5. The results were compared with published data of C_L and C_D for the airfoil. The error lies within 5-10 % as compared with the published values of lift, while the corresponding error for drag values is lying between 8-30%. The pressure and velocities are plotted in form of contours. The results show that the lift coefficient and drag coefficient are the function of angle of attack both are increasing with increasing angle of attack whereas the drag coefficient decreasing with increasing Reynolds number with cruising lift. The maximum lift coefficient occurred at 12° for Reynolds number $Re=3 \times 10^6$ and 14° , 16° for Reynolds number 6×10^6 and 9×10^6 .

Keywords— Angle of attack, Airfoil, CFD, Coefficient of lift, Coefficient of drag, Reynolds number.

I. INTRODUCTION

The rapid evolution of computational fluid dynamics (CFD) has been driven by the need for faster and more accurate methods for the calculations of flow fields around configurations of technical interest. In the past decade, CFD was the method of choice in the design of many aerospace, automotive and industrial components and processes in which fluid or gas flows play a major role. In the fluid dynamics, there are many commercial CFD packages available for modeling flow in or around objects. The computer simulations show features and details that are difficult, expensive or impossible to measure or visualize experimentally. This Studies involving investigation of aerodynamic performance of NACA airfoil were carried out using computational approach to examine the associated flow field developments and the resulting aerodynamic forces at different Reynolds number. The numerical studies were conducted using a commercially available computational fluid dynamics package to simulate flow around the airfoils. The airfoil selected for this exploration was a NACA 63018 a 18% thick airfoil. The NACA 63018 airfoil is using for large wind turbine blades and unmanned Aerial vehicles (UAV) [1].

An airfoil is defined as the cross section of a body that is placed in an airstream in order to produce a useful aerodynamic force in the most efficient manner possible. The cross sections of wings, propeller blades, wind mill blades,

compressor and turbine blades in a jet engine, and hydrofoils are example airfoils [2]. The basic geometry of an airfoil is shown in Figure 1. [3]

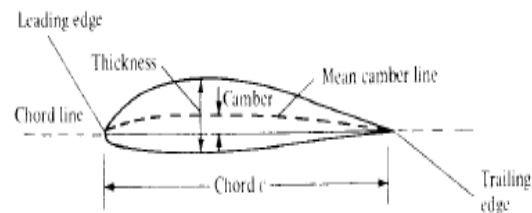


Fig.1 Geometry of an airfoil [3]

The basic geometry of an airfoil is shown in Fig.1. The most important features of airfoil geometry are the chord, camber, Thickness, Mean camber line, Leading edge and Trailing edge. The most forward and rear ward points are called Leading edge and trailing edge. The straight line connecting the leading and trailing edges is the chord line, the distance measured between the trailing and leading edge along the chord line is the chord of an airfoil. The line of points that are halfway between the upper and lower surfaces is the mean camber line as measured perpendicularly from the chord line. The thickness of an airfoil is the distance from the upper and lower surfaces as measured perpendicularly to the chord line, and varies in distance along the chord line. Camber is the maximum distance that occurs between the mean camber line and the chord line [3].

The fig .2 shows the definition of Lift and Drag on an airfoil.

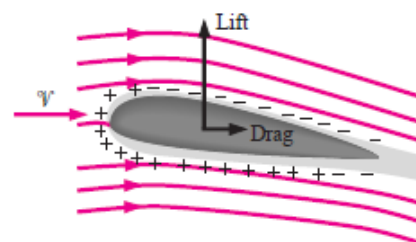


Fig.2 Definition of Lift and Drag [4]

The force a flowing fluid exerts on a body in the flow direction is called drag force F_D .

$$F_D = C_D (1/2) \rho V^2 A \quad (1)$$

The forces in the direction normal to the flow tend to move body in that direction is called Lift force F_L .

$$F_L = C_L (1/2) \rho V^2 A \quad (2)$$

Where, ρ is the density of the fluid in kg/m^3 , V is the velocity of the fluid in m/sec , A is ordinarily the frontal area (the area projected on a plane normal to the direction of flow) of the body, C_D and C_L are the drag coefficient and lift coefficient.

II. CONSTRUCTING THE DOMAIN AND MESHING

The coordinates describing the airfoil were obtained from the UIUC^[5] airfoil database (UIUC, 2007) which described the airfoil by 50 points. The airfoil coordinates were then imported into the computer aided drafting (CAD) program CATIA V5. The coordinates were input as a spline and were broken at the trailing edge of the airfoil. This allowed a sharp trailing edge to be created and the leading edge was set as a continuous part of the spline as shown in fig.3.



Fig.3 Geometry of airfoil created in CATIA V5

After the raw airfoil coordinates had been imported into the CAD program, it was then exported as a Para solid model with extension .stp. The Para solid was then imported into the meshing program ANSYS fluent 14.5. Here the flow field was meshed appropriately.

It was necessary some structured sub divisions of analysis domain to generate the mesh. The domain was discretized by a structured grid of quadrilaterals as shown in fig.4. The finite elements mesh had 135000 nodes and 135900 elements. Different geometry had been constructed for simulation for angle of attack 0° to 16° as shown in fig.5.

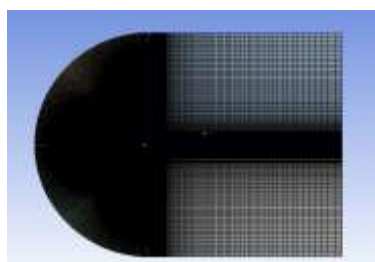
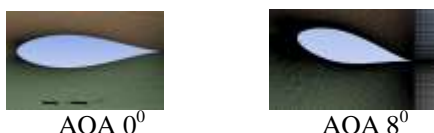


Fig.4 Completed meshes



AOA 0°

AOA 8°

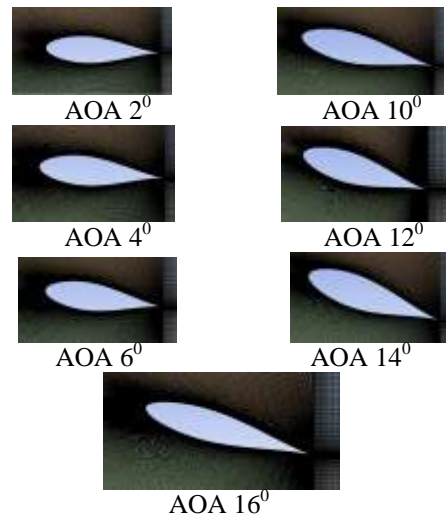


Fig.5 Angle of attack from 0° to 16°

The flow inlet velocities were taken as 45 m/sec , 90 and 130 m/sec in order to match the required Reynolds number $Re=3 \times 10^6$, 6×10^6 and 9×10^6 . The density and kinematic viscosity taken as the standard atmospheric condition 1.225 kg/m^3 and 1.7894e-05 kg/m-s . For all the simulation density based Navier stock (DBNS) with Spalart-Almaras one equation model was used.

III. RESULT AND DISCUSSION

1. Lift and drag coefficients

A numerical investigation of NACA 63018 for various angles of attack at different Reynolds number were done and then validate the result with existing experimental data from reliable sources^[6]. To do so, the model was solved with a range of different angles of attack from 0° to 16° at different Reynolds number $Re=3 \times 10^6$, 6×10^6 and 9×10^6 .

From the tables 1,2 and 3 and fig 6 it observed that the lift coefficient is increasing with increasing Reynolds number at each angle of attack. At Reynolds number 3×10^6 the maximum lift coefficient was 1.26559 at angle of attack 12° . And for Reynolds number $Re= 6 \times 10^6$ and 9×10^6 are 1.4296948, 1.51903 at angle of attack 14° and 16° as shown. Maximum lift coefficient is dependent upon Reynolds number. This makes sense, because maximum lift coefficient is governed by viscous effects^[3], and Reynolds number is a similarity parameter that governs the strength of inertia forces relative to viscous forces in the flow.

It also observed that the drag coefficient increasing with increasing lift coefficient it also a function of angle of attack as shown in the fig.7. There was small variation in drag coefficient at angle of attack 0° - 6° at higher angle of attack transition to turbulent flow occurs over the airfoil surface causing sharp increase in drag coefficient as shown. From the fig.7 it could be observed the drag coefficient is decreasing with increasing Reynolds number. This is because of the drag on any aerodynamic body is composed of pressure drag and skin friction drag^[3]. The basic source of aerodynamic force on

a body are the pressure and shear stress distribution exerted on the body surface .The division of total drag on to its components of pressure and skin friction drag is frequently useful in analyzing aerodynamic phenomena. The NACA 63018 air foil is a stream lined body .The most of the drag of

the stream lined body is due to skin friction. Skin friction coefficient is a function of Reynolds number. It is inversely proportional to the Reynolds number so, drag coefficient decreases as Reynolds number increases.

Table: 1. Comparison of CFD data with NACA’s published data for NACA 63018 airfoil at $Re=3 \times 10^6$

ANGLE OF ATTACK (α)	LIFT COEFFICIENT (CFD)	LIFT COEFFICIENT (NACA)	DRAG COEFFICIENT (CFD)	DRAG COEFFICIENT (NACA)	% OF DEVIATION IN LIFT COEFFICIENT	% OF DEVIATION IN DRAG COEFFICIENT
0°	0	0	0.004230	0.005	0	-15.39
2°	0.23463	0.24	0.004526	0.005	-2.23	-9.47
4°	0.46606	0.48	0.005191	0.006	-2.90	-13.47
6°	0.6936	0.70	0.006391	0.007	-0.91	-8.69
8°	0.91027	0.92	0.011878	0.010	-1.05	18
10°	1.10816	1.10	0.015083	0.016	0.72	-5.72
12°	1.26559	1.23	0.019308	0.017	2	13.5
14°	1.23156	1.25	0.024811	0.022	-1.47	12.7
16°	1.21373	1.22	0.029213	0.028	0.51	4.3

Table: 2. Comparison of CFD data with NACA’s published data for NACA 63018 airfoil at $Re=6 \times 10^6$

ANGLE OF ATTACK (α)	LIFT COEFFICIENT (CFD)	LIFT COEFFICIENT (NACA)	DRAG COEFFICIENT (CFD)	DRAG COEFFICIENT (NACA)	% OF DEVIATION IN LIFT COEFFICIENT	% OF DEVIATION IN DRAG COEFFICIENT
0°	0	0	0.003089	0.004	0	-15.39
2°	0.236430	0.24	0.003384	0.004	-1.48	-15.38
4°	0.469660	0.48	0.003945	0.005	-2.154	-21
6°	0.699632	0.70	0.005996	0.007	-0.05	-14.33
8°	0.924547	0.92	0.010890	0.009	0.49	21
10°	1.125471	1.11	0.012590	0.013	-1.39	3.15
12°	1.300373	1.32	0.017573	0.016	-1.48	9.83
14°	1.429694	1.40	0.028194	0.022	2.12	28.51
16°	1.374600	1.38	0.027053	0.025	-0.39	8.21

Table: 3. Comparison of CFD data with NACA’s published data for NACA 63018 airfoil at $Re=9 \times 10^6$

ANGLE OF ATTACK (α)	LIFT COEFFICIENT (CFD)	LIFT COEFFICIENT (NACA)	DRAG COEFFICIENT (CFD)	DRAG COEFFICIENT (NACA)	% OF DEVIATION IN LIFT COEFFICIENT	% OF DEVIATION IN DRAG COEFFICIENT
0°	0	0	0.00257	0.003	0	-14.33
2°	0.23724	0.24	0.00286	0.003	-1.15	-4.6
4°	0.47144	0.48	0.0034	0.004	-1.78	-15
6°	0.70221	0.70	0.00435	0.006	0.31	-27.5
8°	0.92107	0.92	0.01171	0.010	0.116	17.1
10°	1.14769	1.15	0.01395	0.012	-0.20	16.25
12°	1.30519	1.32	0.0162	0.014	-1.12	15.71
14°	1.4532	1.46	0.02493	0.022	-0.46	13.31
16°	1.51903	1.51	0.02595	0.028	0.59	-7.32

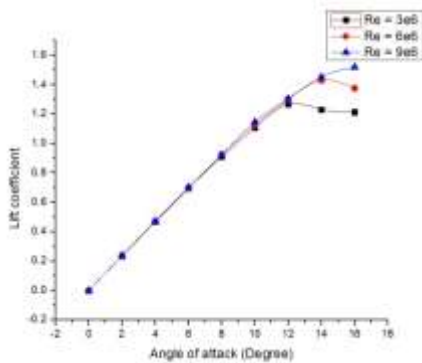


Fig.6 Lift vs Angle of attack for NACA 63018 at Reynolds number $Re=3 \times 10^6$, 6×10^6 and 9×10^6 .

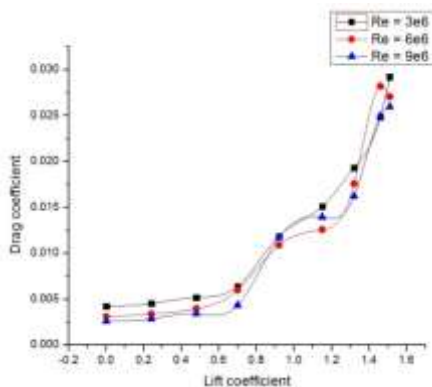
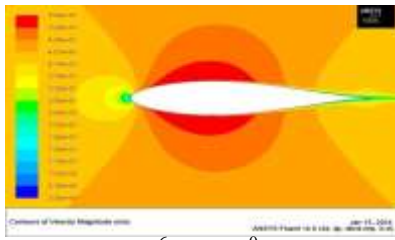


Fig.7 Drag vs Lift coefficient for NACA 63018 at Reynolds number $Re=3 \times 10^6$, 6×10^6 and 9×10^6 .

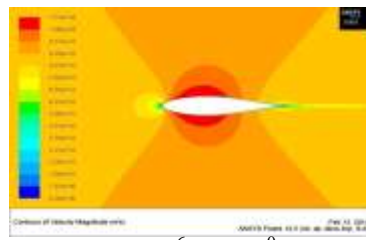
2. Velocity and pressure distribution

From the contours, we see that there is a region of low velocity and high pressure at the leading edge (stagnation point) and region of low pressure and high velocity on the upper surface of airfoil. From Bernoulli equation, we know that whenever there is high velocity, we have low pressure and vice versa [4]. Figure.8 and 9 shows the simulation outcomes of Velocity and static pressure at angles of attack 0° to 16° at different Reynolds $Re=3 \times 10^6$, 6×10^6 and 9×10^6 with Spalart-Aallmaras model. The pressure on the lower surface of the airfoil was greater than that of the incoming flow stream and as a result it effectively “pushed” the airfoil upward, normal to the incoming flow stream.

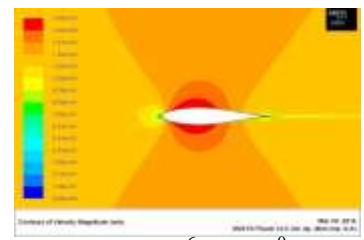
form the figure.9 at angle of attack 0 the maximum pressure and minimum velocity occurs at the leading edge of the airfoil (Stagnation point) and pressure and the velocity at the lower and the upper part of the airfoil is equal .So, there is no lift generation at angle of attack 0° .The stagnation point at the leading edge of the airfoil moves on the lower part of the airfoil with increasing angle of attack, Where the pressure is maximum and the velocity is minimum. When increasing the angle of attack the pressure at the lower surface is increases and velocity decreases.



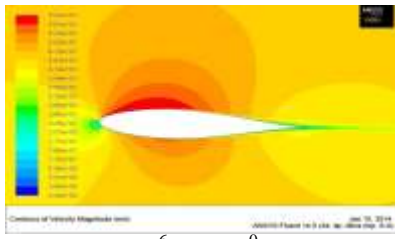
Re= 3×10^6 AOA 0°



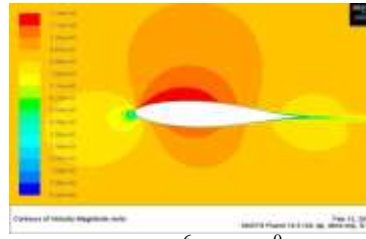
Re= 6×10^6 AOA 0°



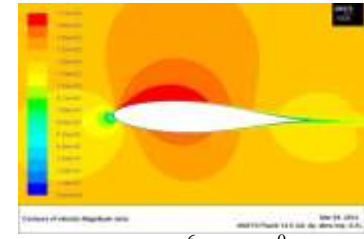
Re= 9×10^6 AOA 0°



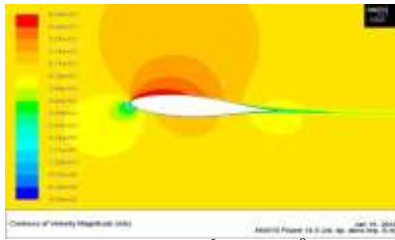
Re= 3×10^6 AOA 2°



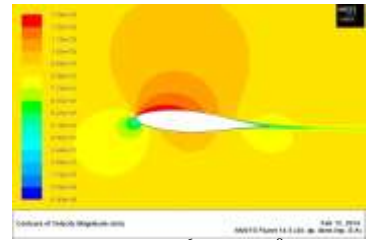
Re= 6×10^6 AOA 2°



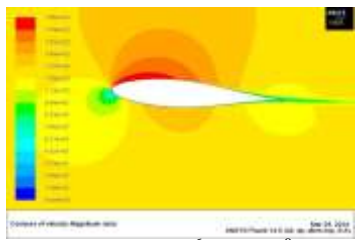
Re= 9×10^6 AOA 2°



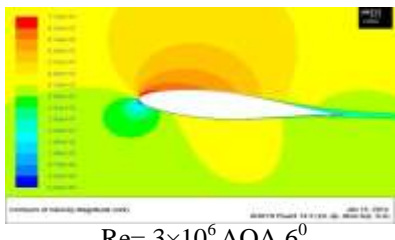
Re= 3×10^6 AOA 4°



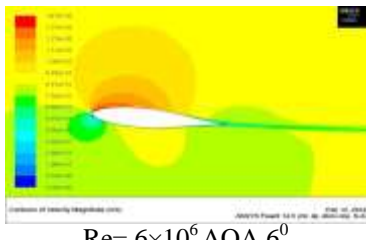
Re= 6×10^6 AOA 4°



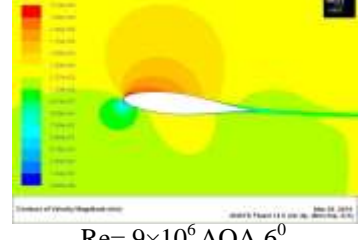
Re= 9×10^6 AOA 4°



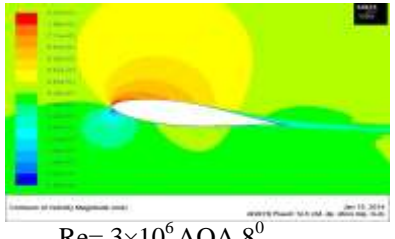
Re= 3×10^6 AOA 6°



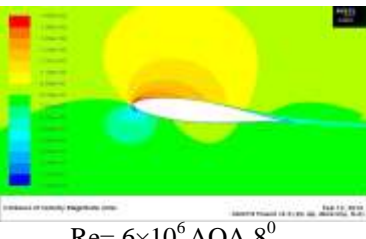
Re= 6×10^6 AOA 6°



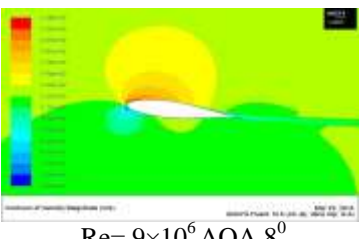
Re= 9×10^6 AOA 6°



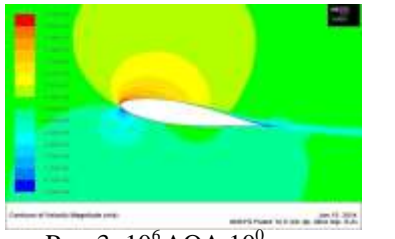
Re= 3×10^6 AOA 8°



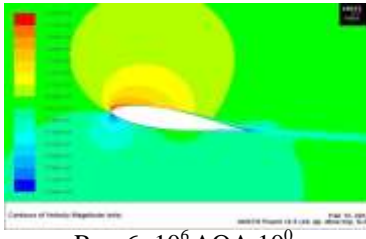
Re= 6×10^6 AOA 8°



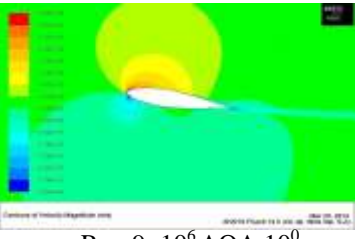
Re= 9×10^6 AOA 8°



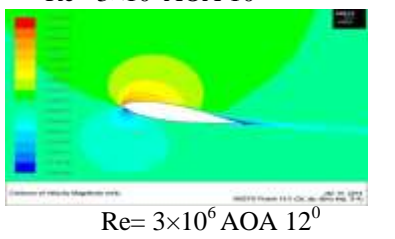
Re= 3×10^6 AOA 10°



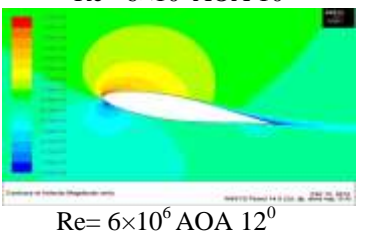
Re= 6×10^6 AOA 10°



Re= 9×10^6 AOA 10°



Re= 3×10^6 AOA 12°



Re= 6×10^6 AOA 12°



Re= 9×10^6 AOA 12°

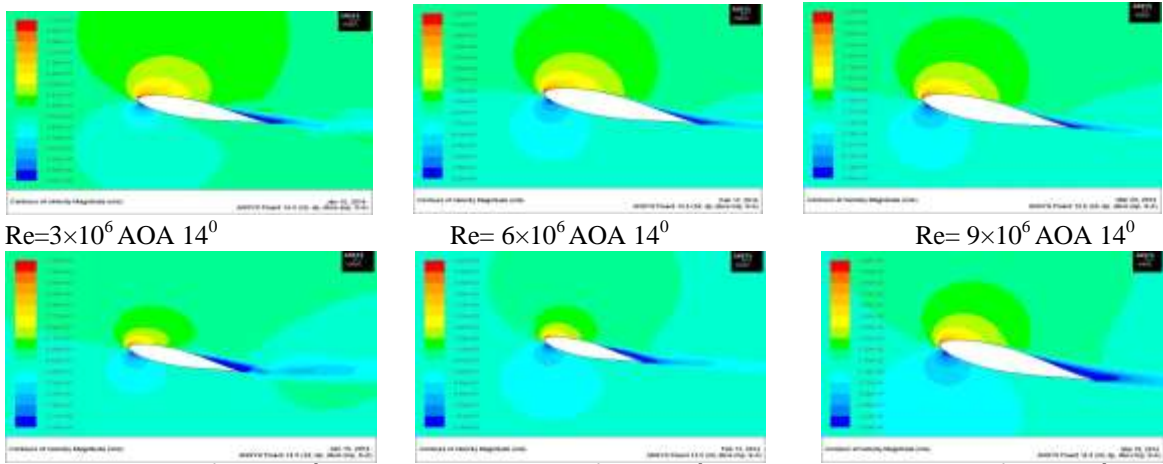
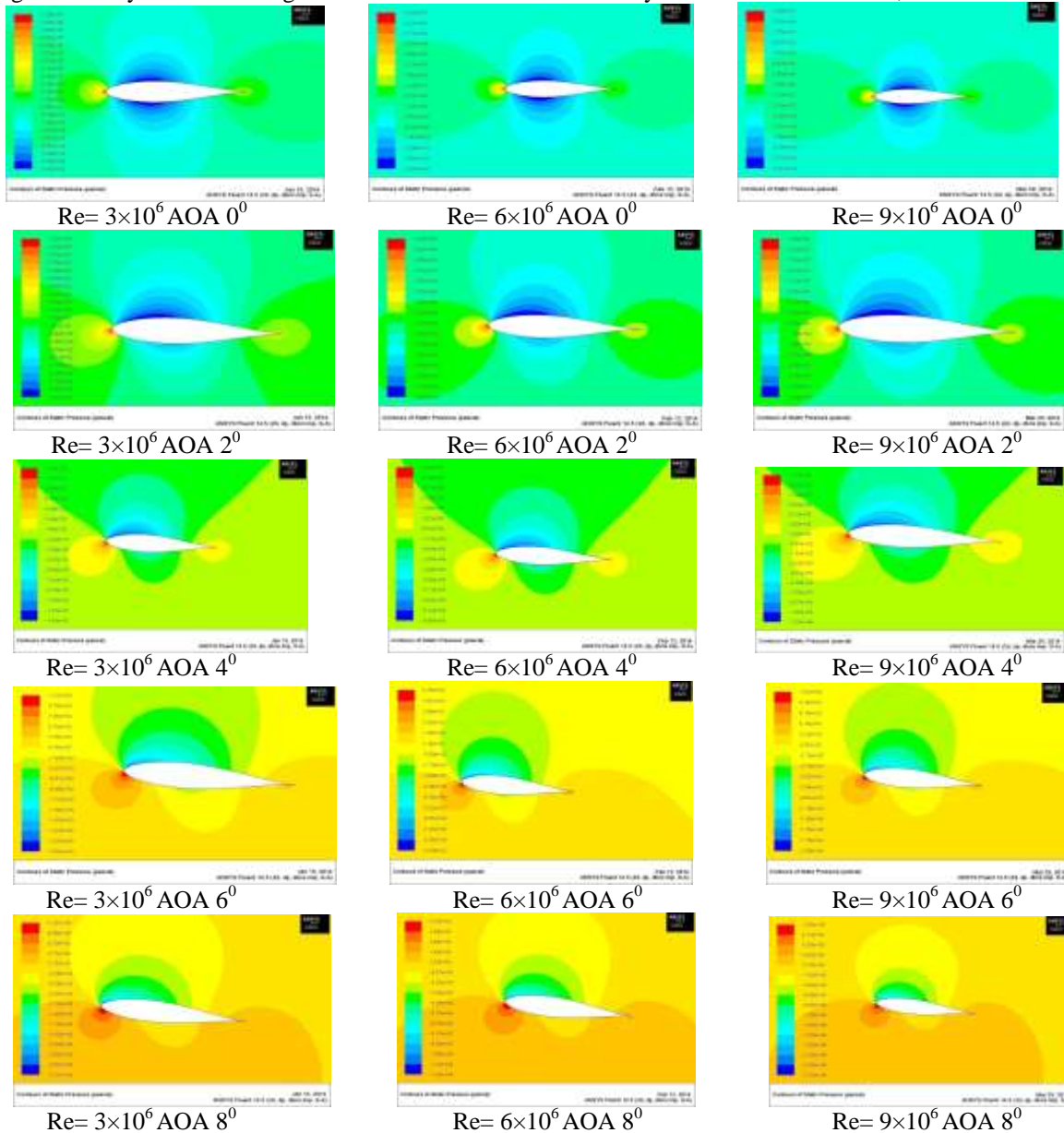


Fig.8 Velocity contours at angle of attack 0⁰ to 16⁰ at different Reynolds number Re=3×10⁶, 6×10⁶ and 9×10⁶.



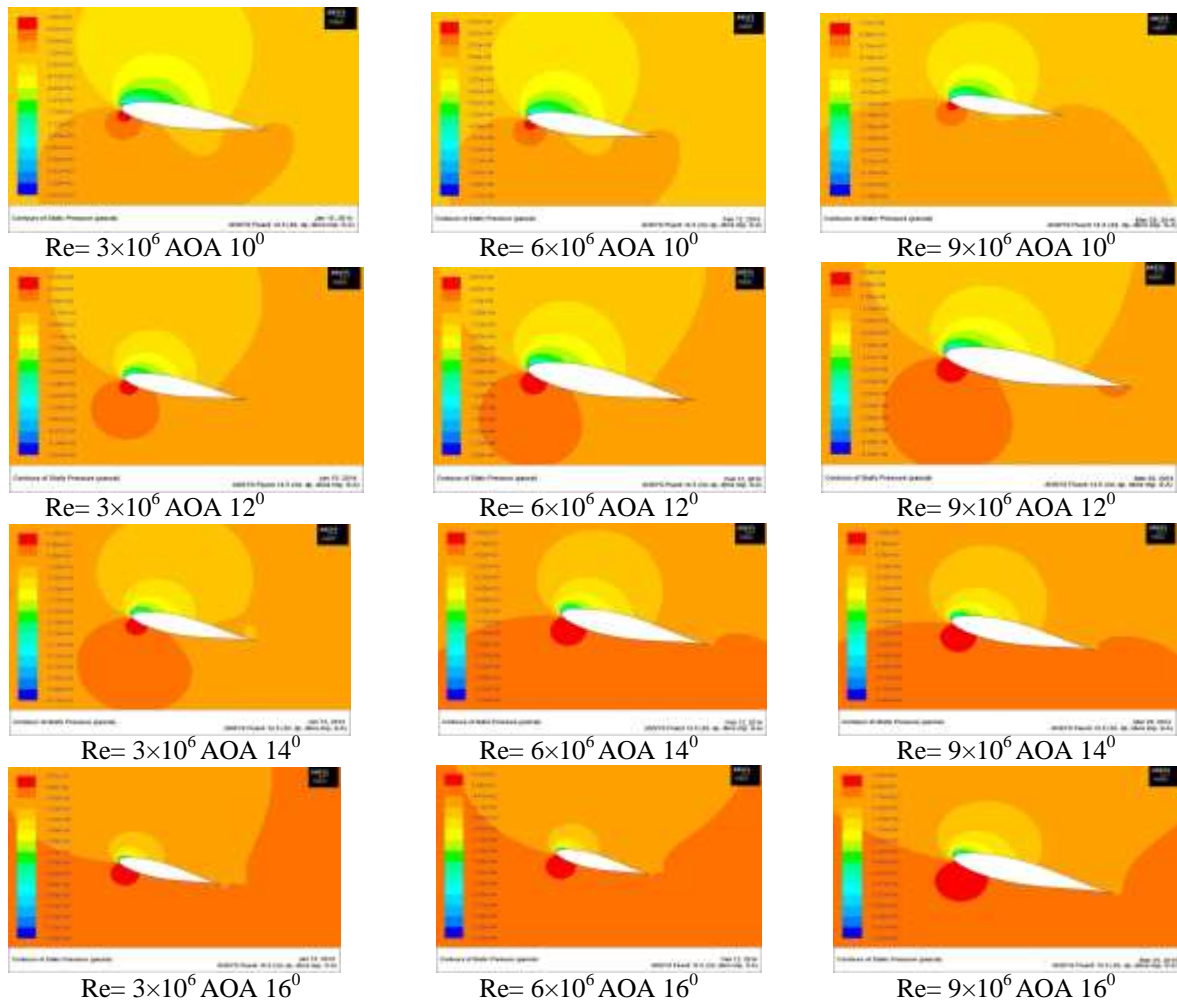


Fig.9 Static Pressure contours at angle of attack 0° to 16° at different Reynolds number $Re=3 \times 10^6$, 6×10^6 and 9×10^6

IV. CONCLUSION

This CFD analysis concluded that the lift coefficient and Drag coefficient are the function of angle of attack. The lift coefficient is increase with increasing angle of attack up to maximum lift coefficient reach and the drag coefficient increases with increasing angle of attack. The drag coefficient decreases with increasing Reynolds number with cruising lift. The maximum lift coefficient occurring at angle of attack 12° for Reynolds number $Re=3 \times 10^6$, and at angle of attack 14° and 16° for Reynolds number $Re=6 \times 10^6$ and 9×10^6 . The error lies within 5-10 % as compared with the published values of lift, while the corresponding error for drag values is lying between 8-30%. From the contours of the CFD analysis of NACA 63018 airfoil conclude that at 0° pressure of upper surface indicate negative pressure. When increase the angle of attack we can understand the decrease the pressure on upper surface and increase on lower surface also became

the maximum at 12° for Reynolds number $Re=3 \times 10^6$, and at angle of attack 14° and 16° for Reynolds number $Re=6 \times 10^6$ and 9×10^6 .

REFERENCES

- [1] W.A. Timmer, Delft An overview of NACA 6- series airfoil characteristics with reference to airfoils for large wind turbine blades, *47th AIAA Aerospace Sciences Meeting Including The New Horizons Forum and Aerospace Exposition* 5 - 8 January 2009,Orland,Florida.
- [2] Pijush K. kundu , Ira M. Cohen, *Fluid mechanics*, Second edition academic press an imprint of Elsevier science , Page no : 629-706
- [3]John.D.AndersonJR,*Fundamentals of Aerodynamics*, Second edition McGraw Hill International.Page no: 247-305
- [4]Yunus cenegal,John M.cimbala, *Fluidmechanics fundamentals and application*, McGraw Hill Page no: 583-632
- [5]http://aerospace.illinois.edu/melig/ads/coord_database.html.
- [6] Ira.H. Abbot,Alberte E.Doenhoff. *NATIONAL ADVISORY COMMITTEE FOR AERONAUTICS Report No. 824*, Summary of airfoil data.
- [7] Scott Richards, Keith Martin, and John M. Cimbala, “ANSYS Workbench Tutorial – Flow Over an Airfoil”, Penn State University, Latest revision:17,January,2011.