Analysis of NACA 4412 and NACA 0018 Airfoil: A Review

Yash Dhaduk[1] Ankitkumar Patel[2] Karnailsingh Saini[3]
[1]B.Tech. Student [2]Head of Department [3]Lecturer
[1,2,3]Department of Aeronautical Engineering
[1,2,3]Parul Institute of Engineering and Technology, Vadodara, India

Abstract— This report is aimed to obtain the velocity and pressure profile, production of Lift and Drag by Airfoil at particular AOA, separation of flow over an Airfoil, the graph of variation in coefficient of lift with AOA (cl-α) and coefficient of Drag with AOA (cd-α) is plotted by using the Ansys Software for an NACA 4412 and NACA 0018 Airfoil. We use fluent module in Ansys for Analysis. CFD plays an important role in design and optimization process. In this project we are comparing the Computational result with the standard result.

Key words: Flow Over an Airfoil, AOA, Aerodynamics, Lift and Drag Force, Coefficient of Lift, Coefficient of Drag, Ansys Fluent

I. Introduction

A. General Information

In earliest day, the only means of transportation was done by walking. But as the technology develops new vehicle for transportation are invented and the latest was air transport which is faster than all. Airplane is use for an air transport which is firstly invented and tested by Wright brothers in 1903. It is most popular after second world war and also used during world war II. After that the air transport led to many new inventions and research to develop faster vehicle [1].

B. Force Acting on an Airplane

When Airplane is soaring in the sky, the four type of Aerodynamic forces acting on it as shown in below figure. They are Lift, Thrust, Drag, Weight. Lift is the upward force which lifts the aircraft for the ground. Thrust is produced by using engine which give forward motion to the aircraft. Drag is in opposite direction of thrust which is the resistance force between air and Aircraft body. Weight is gravity force acting in downward direction due to mass of the aircraft. In which the upward force lift is generated with the help of wing. A wing has an cross section of Airfoil shape. Airfoil is defined as the body which, when placed in a air stream results into occurrence of an aerodynamic forces which can be utilized depending upon the application areas. The Aircraft wing, Aircraft Tail, Helicopter blades, Propeller blades, Compressor blades, turbine blades and fan blades has an airfoil cross-section. Airfoil is first designed by NACA (National Advisory Committee for Aeronautics).

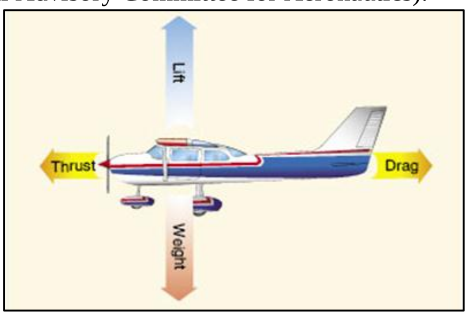

Fig. 1: Forces acting on an airplane

When Airfoil moves in atmosphere the free stream will split into two Components. One will flow on above surface of the airfoil which has high velocity and another will flow below the airfoil which has low velocity. And because of this the pressure difference will found over an airfoil and the lift force is produced by the airfoil.

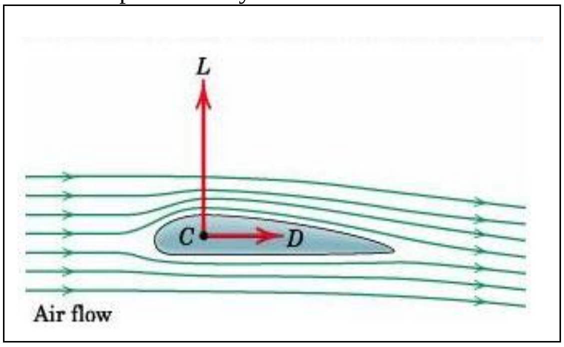

Fig. 2: Lift and Drag over an Airfoil

This lift force is in a normal direction to flow velocity.

The equation of lift is

$$\text{lift} = C_L \times \left(\tfrac{1}{2}\rho V^2\right) \times S$$

And due to this Lift; drag is also be developed and which oppose the motion of an Aircraft. This drag force is in a parallel direction to the flow velocity.

$$\text{drag} = C_D \times \left(\tfrac{1}{2}\rho V^2\right) \times A$$

C. Nomenclature of airfoil

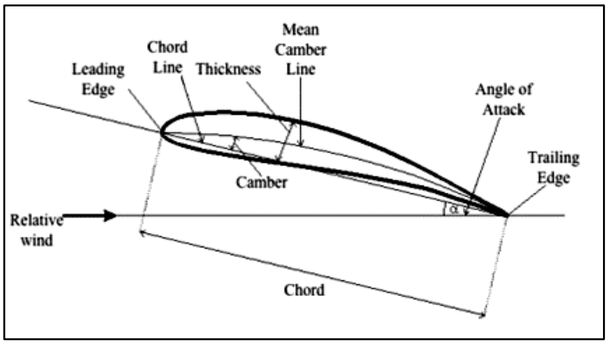

Fig. 3: Nomenclature of Airfoil

Consider the Airfoil sketched in figure.

1) LEADING EDGE: The most forward point of mean camber line is called leading edge.
2) TRAILLING EDGE: The most rearward point of the mean camber line is called trailing edge.
3) CHORD LINE: The straight line connecting the leading and trailing edge is called chord line of the airfoil.
4) MEAN CAMBER LINE: It is the locus of points half way between the upper and lower surface as measure perpendicular to the mean camber line itself.
5) CHORD LENGTH: The precise distance from the leading to trailing edge measured along the chord line is simply designated the chord length C of the airfoil.

6) CAMBER: The camber is the maximum distance between mean camber line and the chord line, measured perpendicular to chord line.
7) THICKNESS: It is the distance between upper and lower surface, also measured perpendicular to the chord line.
8) ANGLE OF ATTACK: It is the angle between the incoming air or relative wind and chord line [2].

D. Types of Airfoil

Basically there are two types of Airfoil.

1) Cambered Airfoil: In cambered airfoil; the Mean camber line and chord line are not coincide to each other. There is some distance between them which is known as camber.
2) Symmetrical Airfoil: In Symmetric airfoil both Mean camber line and chord line coincide to each other.

E. NACA series

NACA has classify the the Airfoil into several series. The first family of NACA was developed in 1903 [2].

Classification of NACA series are:

1) NACA 1digit series
2) NACA 4 digit series
3) NACA 5 digit series
4) NACA 6 digit series
5) NACA 7 digit series
6) NACA 8 digit series

NACA 4412 Airfoil		
Sr. No.	Digit Number	Characteristics
1.	4	4% is maximum camber in percentage of chord
2.	4	40% is the location of maximum camber in percentage of chord
3.	12	12% is the maximum thickness in percentage of chord

Table 1: Geometry parameters of NACA 4412 airfoil

NACA 4412 Airfoil		
Sr. No.	Digit Number	Characteristics
1.	0	It has a no camber because it is symmetric airfoil
2.	0	It has a no camber so location of maximum camber is zero
3.	18	18% is the maximum thickness in percentage of chord

Table 2: Geometry parameters of NACA 0018 airfoil

F. Airfoil Characteristics

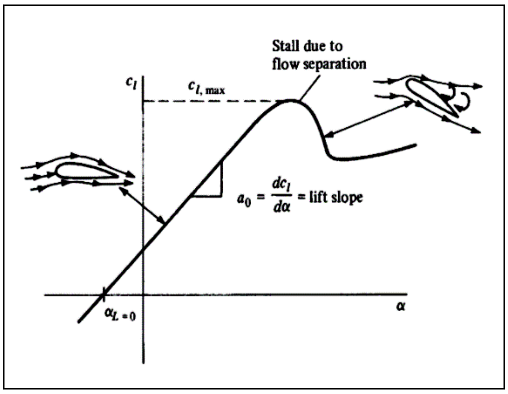

Fig. 4: Lift Coefficient variation with angle of attack for cambered airfoil

The variation of C_l (Coefficient of lift) with α for an airfoil is shown in the above figure. At low to moderate angle of attack, C_l varies linearly with α; the slope of this straight line is denoted by a_0 and it is called as lift slope. In this region, the flow moves smoothly over the airfoil and is attached over most of the surface, as shown in streamline picture at the left. As α becomes large, the flow tends to separate from the top surfaces of the airfoil, creating a large wake of relatively "dead air", which is shown at the right of the figure. In this separated region, the flow is recirculating and part of the flow is actually moving in a direction opposite to the freestream which is called reversed flow. The result of this separated flow at high α will decrease the lift and produces large amount of drag. Under such conditions the airfoil is said to be stalled. The maximum value of C_l, which occurs just prior to the stall, is denoted by $C_{l,max}$. $C_{l,max}$ is the most important aspect of airfoil performance, because it determines the stalling speed of an airplane. The higher is $C_{l,max}$ the lower is stalling speed. After examining the figure we can say that C_l increases linearly with α until the flow separation begins to have an effect. Then the curve become non-linear, C_l which is a maximum value and the airfoil stalls. At the other extreme of the curve, as shown in figure, the lift at α=0 is finite; indeed, the lift goes to zero only when the airfoil is pitched at some negative angle of attack because this is for cambered airfoil. So the value α when lift is equals to zero is called the zero lift angle of attack and it is denoted by α_l=0. For symmetric airfoil, α_l=0 will be zero, where as for all airfoils with positive camber (camber above the chord line), $\alpha_{l=0}$ is a negative value, usually of the order of -2 to-3 degree [2]. Coefficient of lift is given by the,

$$C_l = \frac{L}{\frac{1}{2}\rho V^2 S} = \frac{2L}{\rho V^2 S} = \frac{2L}{qS}$$

And coefficient of drag is given by,

$$C_d = \frac{D}{\frac{1}{2}\rho V^2 S} = \frac{2D}{\rho V^2 S} = \frac{2D}{qS}$$

G. Computational Fluid Dynamics (CFD)

CFD is a branch of Fluid mechanics that uses numerical analysis and data structure to analyze and solve problems that involve fluid flow. CFD is applied to a wide range of research and engineering problems in many field of study and industries, including aerodynamics and aerospace analysis, industrial system design and analysis, engine and combustion analysis. It is used for analysis of fluid mechanics and dynamics problems. Fundamental basis of almost all CFD problems is the Navier-Stokes equations [3]. ANSYS is the majorly used analysis software in the world due to its fast and accurate result. It will offer the engineering simulation solution sets in engineering problem that a design process requires. ANSYS uses FEM and various other programming algorithms for simulating and optimizing various design problems. ANSYS has many modules out of which ANSYS Fluid Flow fluent and Structural are used for fluid flow analysis and structural analysis. ANSYS Fluent contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications.

II. Review of Literature

Jasminder Singh et.al.[(4)] had studied the effect of fluid flow over the NACA 4412 and NACA S1223 through Computational Fluid Dynamics (CFD), Analysis of this two airfoil is done in this from 0 degree Angle of Attack (AOA) to 15 degree AOA with an increment of 5 degree with a velocity of 2.5m/s using the laminar model. And shown the effect of variation in Angle of Attack on velocity profile, pressure profile, coefficient of lift and coefficient of drag. They have compare the result of C_l and C_d of both the airfoil, And conclude that NACA 4412 airfoil has low lift and drag coefficient. Thus it is suitable for sport plane. Sport plane has to cruise to high velocity as compare to heavy lift cargo plane. Therefore drag force should be less, achieved by low drag coefficient [4].

NACA 4412 airfoil				S1223 airfoil		
Angle of Attack (Degree)	cl (coefficient of lift)	cd (coefficient of drag)	cl/cd	cl	cd	cl/cd
0	0.22	0.05	4.4	0.82	0.09	8.28
5	0.67	0.07	9.57	1.26	0.15	8.07
10	0.88	0.13	6.47	1.70	0.25	6.84
15	1.25	0.22	5.58	2.13	0.38	5.61

Fig. 5: Comparison of NACA4412 and S1223 airfoil

Mayurkumar Kevadiya et.al.[(5)] had studied and done the 2D analysis of NACA 4412 airfoil by using Fluent 6.3.26 at various AOA from 0º to 12º. They had design the airfoil by using GAMBIT 2.4.6. They obtained the velocity and pressure profile over an airfoil for different AOA and have calculate the coefficient of lift and drag for 1×10^5 Reynolds no. with an Inlet velocity of 18m/s and using splart allarmas model. Their study of NACA 4412 was primarily focuses on designing of blade for the region of low wind power density. Because the aerodynamics airfoils of wind turbine blades have crucial influence of aerodynamic efficiency of wind turbine. This involve the selection of a suitable airfoil section for the proposed wind turbine blade. And they conclude that C_l/C_d ratio increases with AOA upto 8º. After 8º L/D ratio decreases as AOA increases [5].

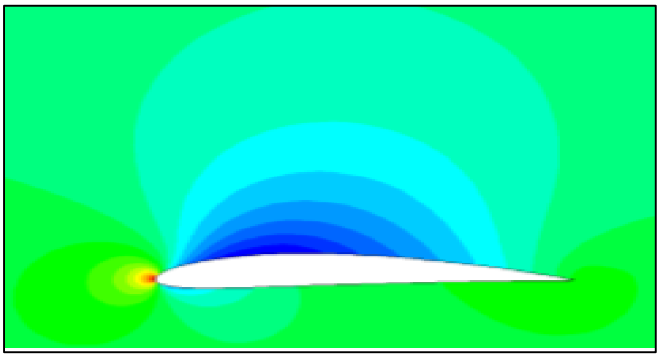

Fig. 6: Pressure contour of NACA 4412 at zero AOA

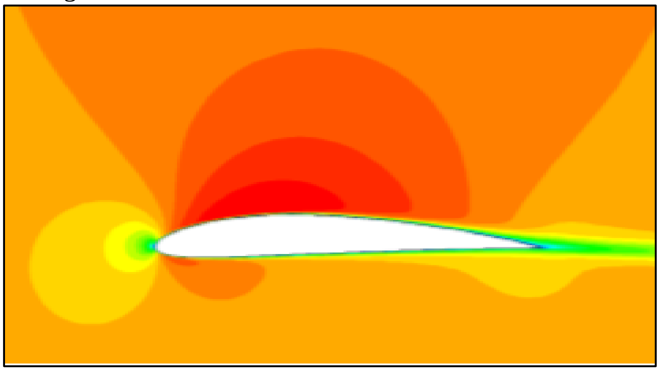

Fig. 7: Velocity contour of NACA 4412 at zero AOA

Karna S, Patel et.al.[(6)] had obtained the life and drag force through analysis by using CFD software for 2D subsonic flow over a NACA 0012 airfoil at various AOA with 3×10^6 Reynolds no. In their study the velocity of the flow is 51m/s with transition steady state (4th equation). They conclude that at 0º AOA their is no lift force generated and if AOA increases the amount of lift force increases [6].

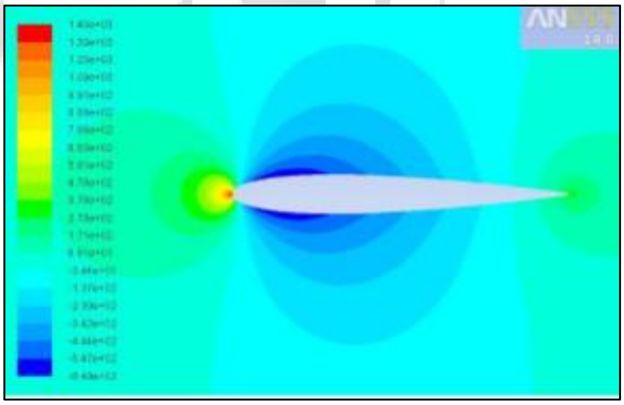

Fig. 8: Pressure contour of NACA 0012 at zero AOA

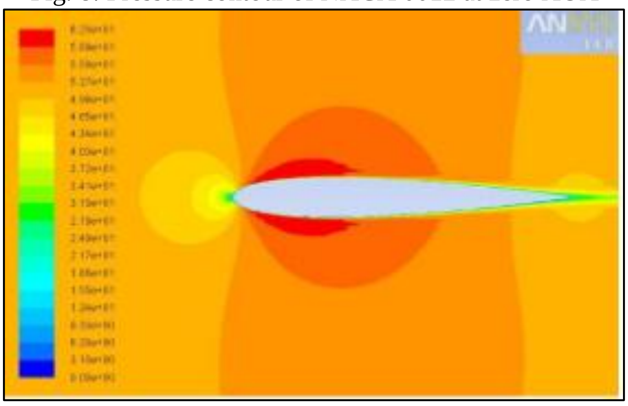

Fig. 9: Velocity contour of NACA 0012 at zero AOA

D, Vimal Chand et.al.[7] had investigate the flow over multi-element of NACA 4412 and NACA 0012 airfoil in Ansys Fluent. For this analysis they have selected 2D viscous transient flow, pressure model with k-ω turbulence model were applied for this analysis, Through this they study properties over this airfoil [7].

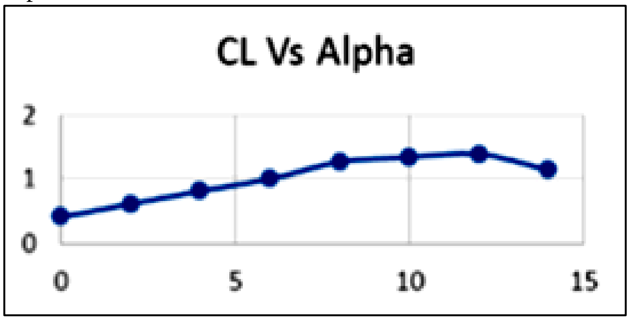

Fig. 10: C_l vs Alpha Curve for NACA 4412

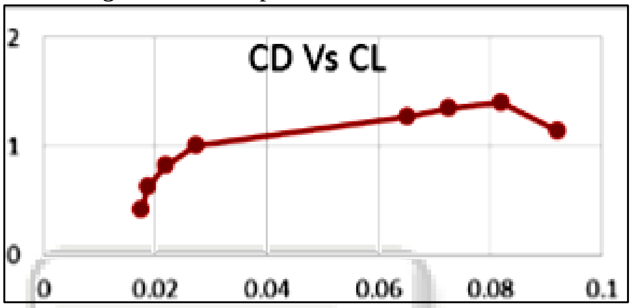

Fig. 11: C_d vs Alpha Curve for NACA 4412

Nguyen Minh Triet et.al.[8] has presented the modeling and simulating process of CFD problem on a Aircraft wing model, using typical section of NACA 2412 airfoil. They uses the Ansys Fluent to analyses the pressure and velocity distribution over wing surface and also determine the lift and drag force over a wing. The coefficient of lift and coefficient of lift drag over a airfoil for 0 to 50m/s velocity is also calculated. And conclude that the lift force is larger about 22,5 times than drag force which allows the lifting of weight of a flying object [8].

S. Kandwal et.al.[9] had presents the computational investigation of inviscid flow over an NACA 4412. The analysis had been done by using Ansys Fluent (version 12.0,16) to obtain the surface pressure distribution from which lift and drag were calculated. In addition to this coefficient of lift and drag were also determined, The fluid used for this is air. And compare this CFD simulation result with experimental data which is taken from theory of wing sections by Abbott et al, And conclude that pressure coefficient is maximum on surface of airfoil, the velocity of upper surface is faster then velocity on lower surface, and their lift and drag computation were found in close agreement with the experimental values [9].

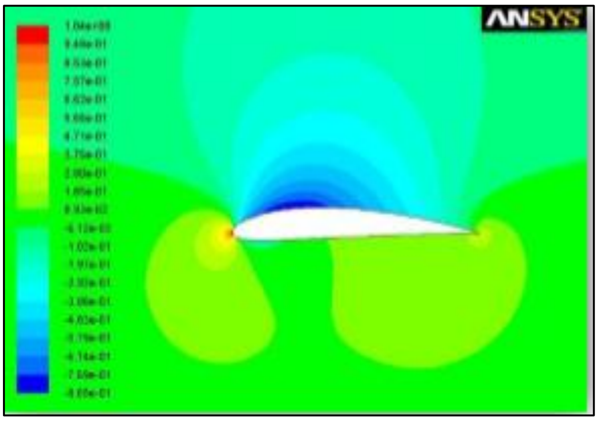

Fig. 12: Pressure contour of NACA 4412 at zero AOA

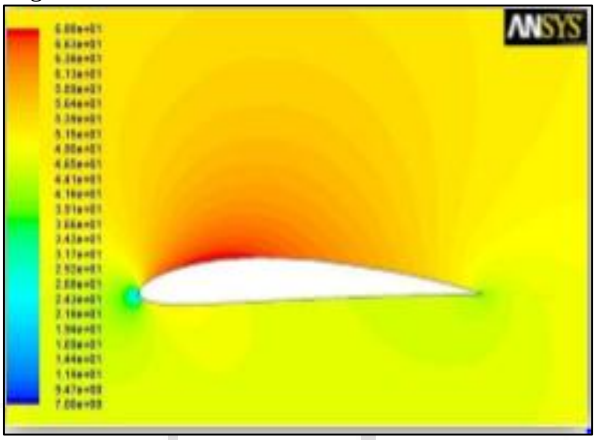

Fig. 13: Velocity contour of NACA 4412 at zero AOA

Abhay Sharma et.al.[10] had generated NACA 4413 airfoil as per Input parameters of NACA airfoil generator, Some of main behavior of airfoil have been examined using CFD software with the flow velocity 2m/s in no slip condition. Same has been verified in real life flow in experimental setup of wind tunnel. With the variation in AOA they have measured the change in velocity profile, pressure profile, flow separation, wake formation in both ways. From real time flow and convergent solution in CFD it has been found that velocity decreases at lower surface and wake formation increases with increase in AOA [10].

REFERENCES

[1] J.E. Copper, "Towards Faster and Safer Flight Flutter Testing", Proc. Symp. Reduction of Military Vehicle Acquisition Time and Cost through Advanced Modelling and Virtual Simulation, Paris, 2002.
[2] Fundamental of Aerodynamics by John D. Anderson Jr.
[3] https://en.wikipedia.org/wiki/Computational_fluid_dynamics
[4] Jasminder singh, Dr. Jaswinder singh, Ampritpal Singh, Abhishek Rana, Ajay Dahiya, "Study of NACA 4412 and Selig 1223 airfoils through computational fluid dynamics", SSRG International Journal of Mechanical Engineering (SSRG-IJME), Vol.2, Issue 6,June 2015
[5] Mayurkumar Kevadiya, Hemish A. Vaidya, "2D Analysis of NACA 4412 Airfoil", International Journal of Innovative Research in Science, Engineering and Technology Vol. 2, Issue 5, May 2013

[6] Karna Patel, Saumil B. Patel, Utsav B. Patel, Prof. Ankit P. Ahuja, “CFD analysis of an Aerofoil” , Internationaal Journal of Engineering and Research, Volume No.3, Issue No.3, 01 March 2014, pp:154-158.
[7] D Vimal Chand, R. Sriram, D, Udaya Kumar, “Aerodynamic analysis of Multi element airfoil”, International Journal of Scientific and Research Publications, Volume 6, Issue 7, July 2016.
[8] Nguyen Minh Triet, Nguyen Ngoc Viet, Pham Manh Thang, “Aerodynamic analysis of aircraft wing”, VNU journal of science, Vol. 31, No. 2 in 2015.
[9] S. Kandwal and Dr. S. Singh, “Computational fluid dynamics study of fluid flow and aerodynamic force on an airfoil”, Internationaal Journal of Research and Technology, Volume 1, Issue 7, September – 2012.
[10] Abhay Sharma, Manpreet Singh Sarwara, Harsimranjeet Singh, Lakshya Swarup, Rajeev Kamal Sharma, “CFD and Real time analysis of unsymmetrical airfoil”, IJRMET, Vol.4, Issue 2, Spl-2 May-October 2014.