Study of CFD Analysis of Conventional Aircraft Wing

Monica Nagawade[1] ,Shubhashree Munot[2],Dattatray Lawate[3] ,Pritesh Patil[4],Tejas Patil[5]

[1245]*Department of Mechanical Engineering ,Smt.KashibaiNavale college of Engineering,Pune*

[3]*Assistant Professor,Department of Mechanical Engineering ,Smt.KashibaiNavale college of Engineering, Pune*

***Abstract*:Aerodynamic problems in general are often difficult to solve by analytical method. Experimental or numerical simulation can be used to analyze the computational models. However, due to the large expenses required in the experimental method, the numerical method is more preferred. As Computational fluid dynamics (CFD) analysis of an aircraft plays a vital role in providing an ideal design of the aircraft, study of the method itself is carried through this review paper.Preliminarily , the evaluation of how Computational Fluid Dynamics (CFD) package may be incorporated into a conceptual design method is studied in this paper. The repeatability of the CFD solution as well as the accuracy of the calculated aerodynamic coefficients is learned through this study. Study is focused on wing design specifically and how the qualitative and quantitative characterization of the wing provides useful information to verify the wing selection and design prior to the time consuming fabrication of aircraft. Minute changes affecting flow characteristics of aircraft , comparison for CFD analysis carried out at various angles of attack for various wing geometries and determination of lift, drag, stall angle and lift-to-drag ratio wing is learned through this study.**

Keywords: Airfoil basics , wing planforms , CFD analysis ,Wind tunnel test.

I. INTRODUCTION

Any aircraft design begins with the conceptual phase, where possible designs are first imagined and evaluated from initial design requirements. In this phase, the designer has the greatest flexibility in determining the layout and configuration of the aircraft. After the conceptual phase, however, only minor changes to the aircraft configuration may occur. Therefore, it is important to have accurate drag and lift predictions early in the design phase when major configuration changes can occur. The accuracy of these predictions must be balanced, however, with calculation speed. This is needed so many types of configurations can be compared and so size optimization on a selected configuration may occur.

The shape and design of an aircraft can dramatically influence how the aircraft is handled and controlled. The stability and control derivatives are essential for flight simulation and handling qualities. Wind tunnel test are a common method that results in derivatives that are highly accurate. The problem with wind tunnel tests is that they are very expensive and can be extremely time consuming. Also, the wind tunnel models are scaled down to fit in the tunnel, and this can have a dramatic change on the results, since the results do not always scale up as easily. Air will flow over a smaller body differently than a larger body, due to the changes in flow characteristics.

Over the past couple of decades, computer simulation has become much more prevalent. Current CFD software’s are is much more accurate than they once were and are is becoming more user-friendly, but still the mesh generation for a model can be difficult and requires a great deal of experience. These software’s can be used over and over again. Also, many different test cases can be run to determine flying qualities in various situations. There are also several different programs that are readily available that can produce highly reliable results. With the use of CFD, the cost and time required for prototyping has reduced drastically over the years, thereby accelerating the design and development of modern aircrafts.

II. STUDY OF AN AIRFOIL

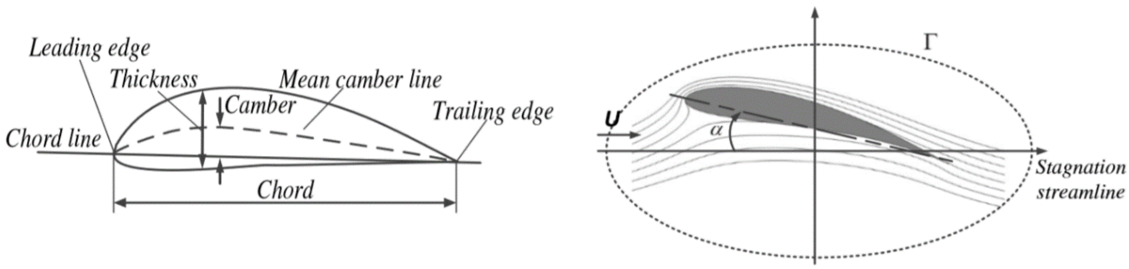

Fig 1 :Basic nomenclature of an airfoil and lift generating process of an airfoil[1]

An airfoil is a body of such a shape that when it is placed in an airstreams , it produces an aerodynamic force. This force is used for different purposes such as the cross sections of wings, propeller blades, windmill blades, compressor and turbine blades in a jet engine. [1]

The leading edge is the point at the front of the airfoil that has maximum curvature. The trailing edge is defined similarly as the point of maximum curvature at the rear of the airfoil. The chord line is a straight line connecting the leading and trailing edges of the airfoil. The chord length, or simply chord is the length of the chord line and is the characteristic dimension of the airfoil section. Angle of attack (AOA) is the angle between the oncoming air or relative wind and a reference line on the airplane or wing. Sometimes the reference line is a line connecting the leading edge and trailing edge at some average point on a wing. [1]

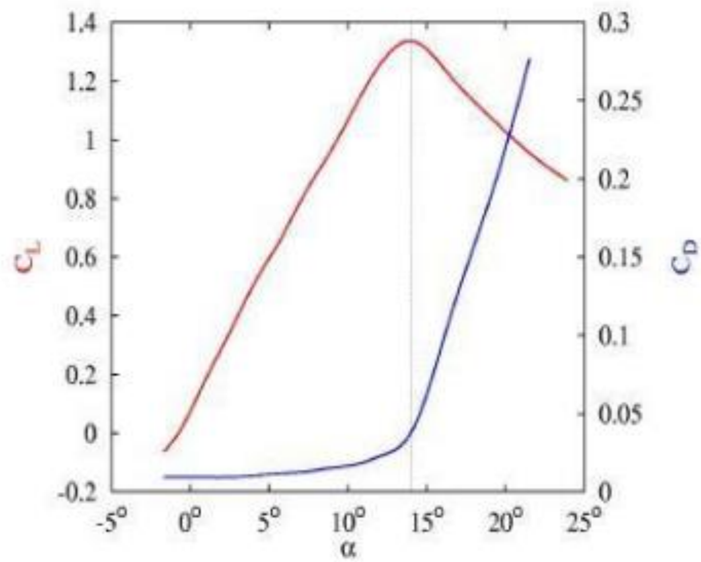

Fig 2: Relationship between angle of attack, coefficient of drag and coefficient of lift[1]

III. STUDY OF NACA 4412 AND SELIG 1223 AIRFOILS

From this paper[2] , we studied the effect of fluid flow over 2 airfoils 4412 and S1223,which are generated as per the NACA (national advisory committee for aeronautics) standards. The comparison was done on the basis of coefficient of lift and coefficient of drag. The angle of attack was varied and their effect was seen on velocity, pressure, coefficient of lift and coefficient of drag. It also helped in specifying the application area of both the airfoils. In case of NACA 4412 airfoil, drag and lift coefficient are low. Thus it is suitable for use in sports planes. The sport plane has to cruise to high velocity as compared to the heavy lift cargo planes. Therefore the drag force should be less, achieved by low drag coefficient. Because of the high velocity the airfoil is able to maintain the lift force even in the case of low lift coefficient. In the case of S1223 airfoil, drag and lift coefficient are high .Thus it is suitable for heavy lift cargo planes. The lift force requirement is higher in cargo planes as it has to lift heavy loads. This is achieved by high lift coefficient. The velocity requirement in cargo planes is lesser as compared to sports plane. Due to lesser velocity the drag force isless even in case of high drag coefficient.

S1223 airfoil		
S.No	Terminology	Characteristics
1.	Maximum camber	8.1% of chord length
2.	Position of maximum camber	49% of chord length
3.	Maximum thickness	12.1% of chord
4.	Position of maximum thickness	19.8% of chord

Table I: Characteristics of S1223 Airfoil[2]

NACA 4412 airfoil		
S. 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 II : Characteristics of NACA 4412 Airfoil [2]

IV. STUDY OF SELIG 1223

The paper [3]focusing on all important factors related to enhanced high lift low Reynold number airfoil performance is studied. This paper developed high lift airfoil design philosophy and validated it through wind tunnel testing. Three codes for airfoil design and analysis (PROFOIL, the Eppler code, and ISES) were used to design the example S1223 high-lift airfoil for a Reynolds number of $2*10^5$. As a result of this work, it is clear that low Reynolds number airfoils can be designed to achieve lift coefficients much higher than previously thought possible. Such high-lift performance can be achieved through the use of a design philosophy that fully exploits the favorable effects of both a concave pressure recovery and aft loading. Surprisingly, as the example airfoil illustrates, the pressure recovery for this class of airfoils, though concave and close to a Stratford distribution, can be tailored to produce acceptable stall characteristics for UAV applications.

V. STUDY OF WING PLANFORMS

From this paper [4] we studied the influence of wing geometry on the aerodynamic performance of micro aerial vehicles (MAVs) from this paper. Four different wing planforms: rectangular, Zimmerman, inverse Zimmerman, and elliptical are considered is this study. The peak of aerodynamic performance moves to lower angles of attacks as the aspect ratio of the wing is increased. The influence of the edge-shape of the wing on the aerodynamic coefficients is also studied. We find that moderate edge-ratio of 2 or 3 produces the lowest drag coefficients. Changing the side-edge geometry influences the lift coefficients but not the drag coefficients.

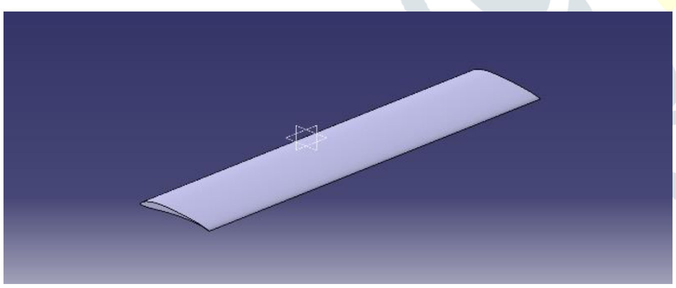

Fig 3 :Rectangular Wing planform

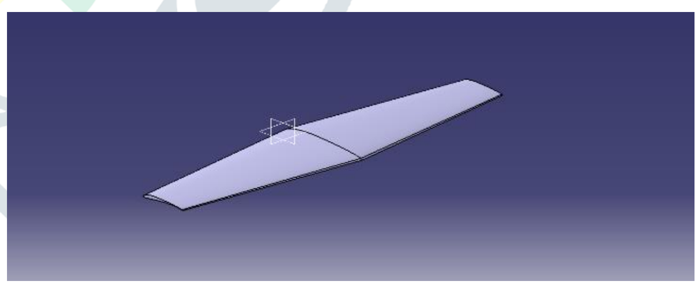

Fig 4 : Tapered wing planform

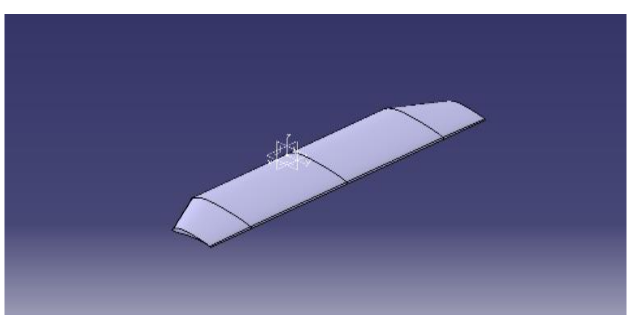

Fig 5 :Semi tapered wing planform

VI. COMPUTATIONAL FLUID DYNAMICS

This paper [5] mainly helped in understanding CFD approach of problem solving and its detailed methodology. It justified why an experimental method is not always feasible and compatible. This paper also provided comparative analysis of two prototypes of wing airfoil. Pressure and velocity contour at various angle of attack were compared for both prototypes. The lift, drag, stall angle and lift-to-drag ratio of each wing were determined and the comparative study of these showed how minute changes to the wing improves its overall flow characteristics. For this very purpose, a virtual wind tunnel model was created and CFD analysis was carried out at various angle of attacks for each wing separately. Analysis played a significant role in improving the wing design of the VTOL/Hybrid Aircraft. Modification was done on the wing after the preliminary analysis, thus shortening the time-cycle before actually coming out with the optimum design. The analysis of the wing helped in significantly predicting a better performance and reduced power consumption of the aircraft due to reduced drag and increased lift forces.

In this paper [9] found the drag and lift forces of aircraft delta wing using CFD. In experimental setup, the design model has to place in testing; this process is quite difficult & cost more than CFD techniques cost for the same. Thus, the entire analysis gone through computational fluid dynamics method. The analysis of the two-dimensional subsonic flow over a NACA 64A206 airfoil at various angles of attack and operating at a Reynolds number of 3.57×E+06 is presented. The CFD simulation results show good agreement with theoretical values it leads to recommend an optimized wing. But every country keeps the details (Information) of the fighter aircrafts are very confidential for safety purpose. In India DRDO is also keeps the all information about its fighter aircrafts. In our project we studied on CFD analysis of delta wing, as we know that delta wings aircraft are either subsonic or supersonic, it gives high speed in normal altitude in this project we studied on two models of delta wing.

VII. WIND TUNNELS AND FLIGHT

In view of the current limitations of predictive analyses, nearly all aircraft designs are tested in wind tunnels in order to accurately estimate performance. These wind tunnel tests are carefully performed to reduce the financial risk of missing the performance.[7] This paper[6] primarily helped in understanding the wind tunnel testing significance , wind tunnel data correction to account for differences between the wind tunnel and flight environments and extrapolating wind-tunnel results to flight conditions.In view of the current limitations of predictive analyses, nearly all aircraft designs are tested in wind tunnels in order to accurately estimate performance. These wind tunnel tests are carefully performed to reduce the financial risk of missing the performance guarantees given to the buyers of the airplane. The open issues are in the geometric fidelity of wind tunnel and computational models relative to flight vehicles, Reynolds number extrapolation, and effects of wind-tunnel flow quality.

VIII. BOUNDARY CONDITIONS

Wee Choon Tan [December 2010], This paper[8] majorly discussed the boundary condition for computational fluid dynamics (CFD) software in order to generate the flow pattern which will be used to compared with the visualized flow pattern. Airfoil used in this paper is NACA0012. The boundary conditions having a great effect to the computational fluid dynamics analyze. The project conducted discusses the methodology and it only considers one of NACA profile aerofoil that is NACA 0012. The design of aerofoil is in two dimensional (2-D) shapes to eliminate the vortex phenomena at the tip of aerofoil. In reality, the aerofoil would be exposed to three dimensional (3-D) flows. However, this would greatly increase the number of variables to be explored.

VIII. CONCLUSION

In this paper all the study prerequisite for design ,analysis and experimental testing of a conventional aircraft wing is done through various sources. General process of aerodynamic analysis of the airflow over airfoil can be understood through this. Selecting the appropriate airfoil as per the requirement , selecting wing planform, Computing lift and drag coefficients using the numerical simulation and further comparing the numerical results with experimental testing such crucial steps are need to be understood before performing any research in the field of aerodynamic analysis of aircraft through computer simulation. The analysis of the wing has helped in significantly predicting a better performance and thus reducing the cost of failure after the manufacturing.

REFERENCES

1) 1 .Karna S. Patel, Saumil B. Patel, Utsav B. Patel, Prof. Ankit P. Ahuja UVPCE, Ganpat University (March 2014) ,"CFD Analysis of an Aerofoil " , published in International Journal of Engineering Research ,
2) Jasminder Singh, Dr. JaswinderSingh , School of Mechanical Engineering, Chitkara University, Punjab, India, "Study of NACA 4412 and Selig 1223 airfoils through computational fluid dynamics" , SSRG International Journal of Mechanical Engineering, Volume No.2.
3) Michael S. Selig and James J. Guglielmo , University of Illinois at Urbana- Champaign, Urbana, Illinois 61801 , "High-Lift Low Reynolds Number Airfoil Design" , published in February 2015 , JOURNAL OF AIRCRAFT Vol. 34.
4) Mehran Tadjfar, Amirkabir University of Technology 59 Publications , " Study of Planform – Shape Influence on MAV Wings", published in January 2009.
5) Shreyas Krishnamurthy, SurajJayashankar, Sharath V. Rao, 5Department of Mechanical Engineering, JSS Academy of Technical Education, Bangalore, "CFD Analysis Of An RC Aircraft Wing" , Sept 2014
6) Henry McDonald, James Ross, David Driver, and Stephen Smith ,NASA Ames Research Center Moffett Field, CA, 94035 USA , "Wind Tunnels and Flight " ,published in May 2008.
7) Nguyen Minh Triet*, Nguyen Ngoc Viet, Pham ManhThang ,Falcuty of Mechanics and Automation, VNU University of Engineering and Technology, 144 Xuan Thuy, CauGiay, Hanoi, Vietnam ,"Aerodynamic Analysis of Aircraft Wing", May 2015.
8) Wee ChoonTan , University Malaysia Perlis ,"Computational fluid dynamics study for aerofoil" ,Published in December 2010.
9) Manoj K. Bhardwaj and Rakesh K. KapaniaVirginia Polytechnic Institute and State University, Blacksburg, Virginia 24061-0203 Eric Reichenbach , "Computational Fluid Dynamics/Computational Structural Dynamics Interaction Methodology for Aircraft Wings" , AIAA JOURNAL Vol. 36, No. 12, December 1998.