Aerodynamic Investigation Turbulent Flow over Wing NACA 0012 using ANSYS

Niranjan Sonawane[1], Harshal Patil[2], Omkar Chavan[3], Sarthak Kanthale[4]

[1,2,3,4]*Undergraduate student, Department of Mechanical Engineering, MIT School of Engineering, MIT ADT University Pune, Maharashtra-412201*

Abstract: **–** In the field of aerodynamics and aircraft design key features are always based on performance characteristics of the structure under different act of application whereas the need of optimum design is always preferable to increase aerodynamic efficiency of system. In current proposed paper NACA 0012 Aircraft wing has been investigated by computational means of commercial CFD ANSYS software to determine its performance under implementation of subsonic speed range which deserve between 0.45-0.75 MACH.

Key words - *CFD, NACA-0012, Wing, ANSYS, MACH, Sub-sonic, Optimum design, Fluent, Pressure Difference.*

I. INTRODUCTION

In Airplane design wing is the most important aspect of design since the performance of aircraft is directly coupled to the Aerodynamic performance of the wing. During immense of First world war product invention and development was at its peak whereas there was an large increase in demand of increasing efficiency of existing military products to coup the demand and promote industrial experts, university and scientific researcher federal government of United states established an board advisory Known as NACA (National Advisory Committee for Aeronautics) was developed in late year of 1915. The promoter goal of NACA advisory is to promote the high research fidelity in-between industrial experts and institutional research aspirants in the field of astronautics and aeronautical Engineering. Promoting advancement in different sectors of from NACA duct for the automotive sector the research and development of NACA developed unique series of NACA Airfoils which helps a lot for aspirant making and sensible design which increase aerodynamic performance of the structure. These are the Airfoils having shape of aircraft wing which is having unique cross-sectional area according to the special digit series. There are total two type of digit series which is subdivided in number of sub series. Main digits are –a) 4 Digit series b) 5 Digit series. This digit series are generated by using special type of Analytical equation which get diverse in variation between its curvature, Middle line which is also known as geometric center, Camber positioning and thickness on which different customized standard airfoil can be generated using NACA advisory calculator. In present paper NACA 0012 type of Airfoil is use this type of airfoil is also known as symmetry airfoil because it distribute equal percentage of geometry in divination from mean line. Due to the symmetrical structure this airfoil promotes the ideal working aerodynamic property which means if it is having zero angle of attack airfoil will not promote any type of lift occurrence. Due to this unique aerodynamic behavior and working property many researcher use NACA 0012 airfoil for experimental use to study the lift and drag characteristics affects within the creation of practical working boundary conditions in practical wind tunnels but experimental testing in wind tunnels accrue large occupancy and working cost to fed up that there is an new computational approach whereas by using commercial CFD software like Hypermesh,ANSYS-Fluent etc. we can setup practical working condition to active precise result that can evaluate for research work. In present work ANSYS CFD fluent software is used to investigate NACA 0012 wing which is first priory developed by using NCAC Co-ordinate calculator and surface is created using ANSYS Parametric design modeler.

II. AIRFOIL NOMENCLETURE

Designing of wing is based on the generalized crossection which is very useful for taking lift. This crossedction designing is done by using based of identical airfoil family since wings in advance commercial based aircraft is developed by using different combinations of airfoil. An aerodynamic force is achieved by airfoil when it is moving through fluid. Lift is generated when the force is in perpendicular direction of motion and whereas Drag can be generated when force is acting on parallel direction of motion as shown in figure 1.

Lift on the airfoil is variedly observed by improving its angle of attack (AOA) whereas for generating proper amount of suitable lift it is necessary that it employed with positive angle ofattack.

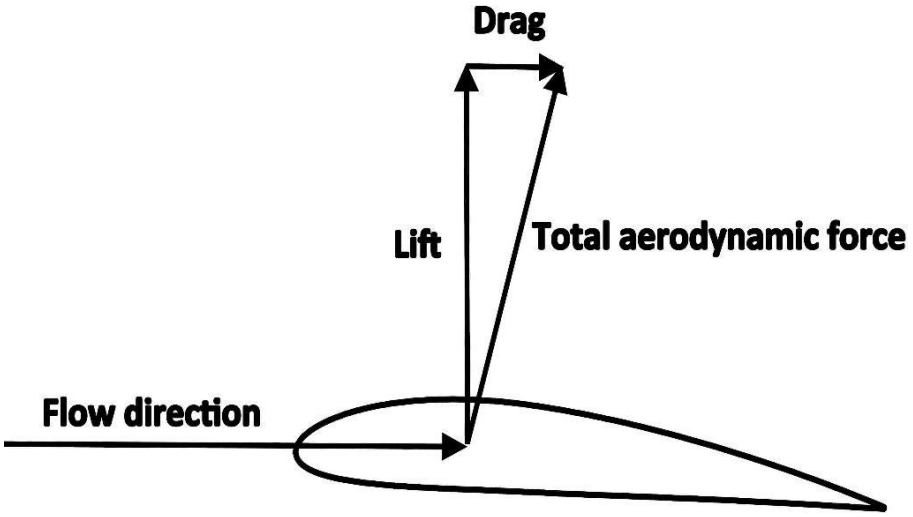


Figure 1. Airfoil Aerodynamic forces.

Nomenclatur es-

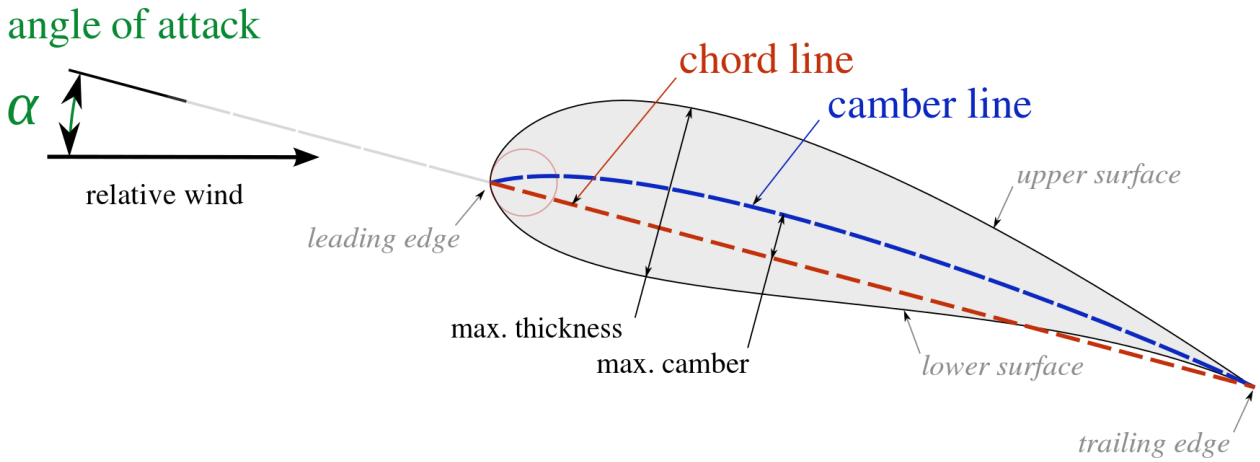
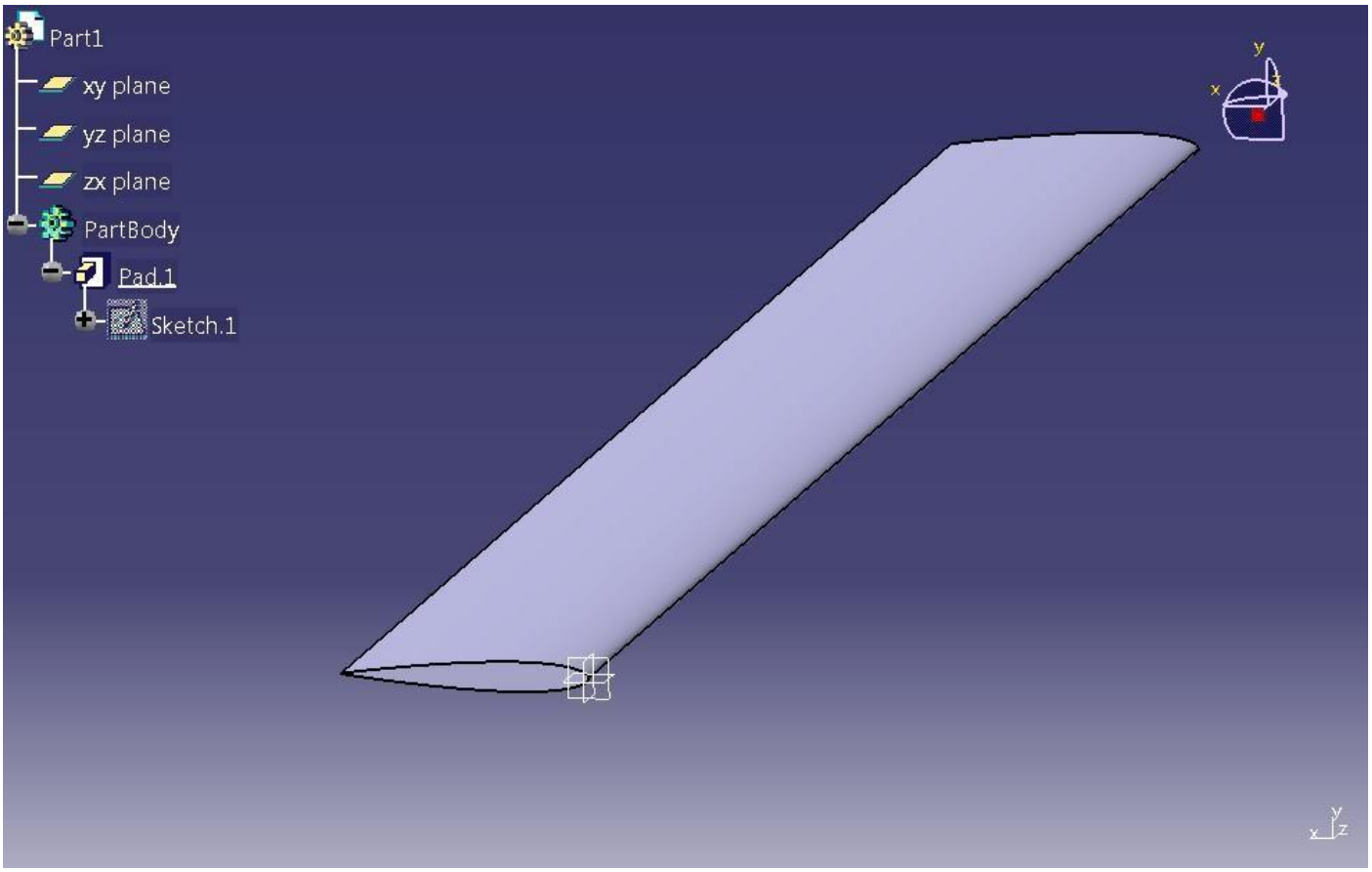


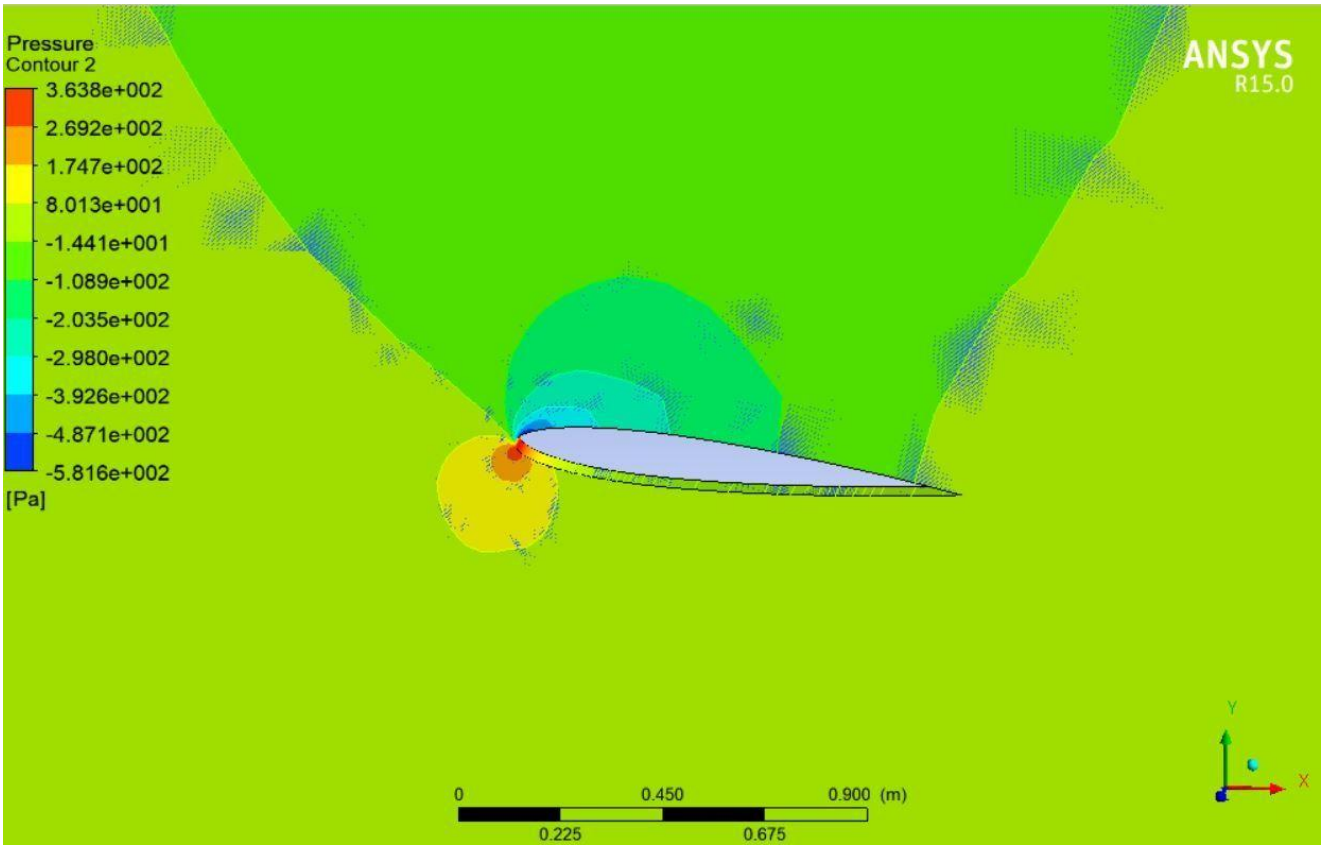
Figure 2. Airfoil Nomenclature.

- The front face of airfoil is known as leading edge which is having maximum curvature and minimum radius.
- The rear point of airfoil is known as trailing edge.
- Leading edge and trailing edge is connected by chord line.
- Alpha represent the degree in Angle of attack.

III. RESULTS AND DISCUSSION

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other related physical processes. It works by solving the equations of fluid flow (in a special form) over a region of interest, with specified (known) conditions on the boundary of that region. Essentially there are three methods for determine the solution to flow problems viz. Experimental, Analytical and Numerical. At present, in order to shorten product development time, there is a strong tendency to perform design using computational fluid dynamics (CFD) tools instead of experiments. CFD is a method that is becoming more and more popular in the modeling of flow systems in many fields, including reaction Engineering. CFD based modeling however has many advantages during preliminary design, because it is less time-consuming than experiments and because it allows greater flexibility. CFD is playing a strong role as a design tool as well as a research tool. In CFD the physical aspects of any fluid flow is governed by three principles. The fundamental equations of fluid mechanics are based on the following Universal laws of conservation: 1. Conservation of mass 2. Conservation of momentum 3. Conservation of energy. These fundamental physical principles can be expressed in terms of basic mathematical equations. These equations are generally in integral or partial differential form. These equations and their derivatives are replaced in CFD by discretized algebraic forms, which are in turn solved to get flow field values at discrete points in space and/or time.


Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
y
x


Pressure
Contour 2
3.638e+002
2.692e+002
1.747e+002
8.013e+001
-1.441e+001
-1.089e+002
-2.035e+002
-2.980e+002
-3.926e+002
-4.871e+002
-5.816e+002
[Pa]
ANSYS
R15.0
0
0.450
0.900 (m)
0.225
0.675
Y
X

IV. CONCLUSION

The following conclusion has been drawn after the analysis of the symmetrical airfoils: 1. the lift and the drag keeps increasing as the AOA increases 2. The portioned increase in the lift is much higher than the drag.

REFERENCES

1. Shyy, W.,Lian ,Y.,Tang ,J.,Liu, H.,Trizila ,P.,Stanford ,B.,Bernal ,L. Cesnik, C.,Friedmann,., Ifju P.: Computational aerodynamics of low Reynolds number plunging, pitching and flexible wings for MAV applications. Acta Mech. Sinica 24, 351– 373 (2008). doi:10. 1007/s10409-008-0164-z.
2. Koochesfahani, M.M.: Vortical patterns in the wake of an oscillating airfoil. AIAA J. 27, 1200–1205 (1989).
3. Triantafyllou, G.S.,Triantafyllou, M.S.and GrosenbaughM.A.: Optimal thrust development in oscillating foils with application to fish propulsion. J. Fluids Struct. 7, 205–224 (1993) 123.
4. Taylor, G.K., Nudds, R.L., Thomas, A.L.R.: Flying and swimming animals cruise at a Strouhal number tuned for high power efficiency. Nature 425, 707–711 (2003). doi:10.1038/nature02000.
5. Read, D.A., Hover, F.S., Triantafyllou, M.S.: Forces on oscillating foils for propulsion and maneuvering. J. Fluids Struct. 17, 163– 183 (2003).
6. Manishaben Jaiswal " SOFTWARE ARCHITECTURE AND SOFTWARE DESIGN" International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056, p-ISSN: 2395-0072, Volume: 06 Issue: 11, s. no - 303 , pp. 2452-2454,
7. Nov 2019 Available at: https://www.irjet.net/archives/V6/i11/IRJET-V6I11303.pdf
8. Manishaben Jaiswal "RISK ANALYSIS IN INFORMATION TECHNOLOGY" , International Journal of Scientific Research and Engineering Development (IJSRED) , ISSN:2581-7175, Vol 2-Issue 6, P110, pp. 857-860, November - December 2019 Available at: http://www.ijsred.com/volume2/issue6/IJSRED-V2I6P110.pdf
9. Manishaben Jaiswal, Mehul Patel "THE LEARNING ON CRM IN ERP- WITH SPECIAL REFERENCES TO SELECTED ENGINEERING COMPANIES IN GUJARAT", International Journal of Management and Humanities Scopus (IJMH) , published by Blue Eyes Intelligence Engineering & Sciences Publication (BEIESP), ISSN 2394-0913, Volume-4 Issue-8, April 2020, Pg-117- 126, Available At, http://www.ijmh.org/wp-content/uploads/papers/v4i8/H0798044820.pdf
10. Hover, F.S., Haugsdal, O., Triantafyllou, M.S.: Effect of angle of attack profiles in flapping foil propulsion. J. Fluids Struct. 19, 37– 47 (2004). doi:10.1016/j.jfluidstructs.2003.10.003.
11. Ramamurti, R., Sandberg, W.: Simulation of flow about flapping airfoils using finite element incompressible flow solver. In: AIAA 37th Aerospace Sciences Meeting and Exhibit, Reno, Nevada, pp. 253–260 (1999).
12. Wu, J.H., Sun, M.: The influence of the wake of a flapping wing on the production of aerodynamic forces. Acta Mech. Sinica 21, 411–418 (2005). doi:10.1007/s10409-005-0064-4.
13. Aono, H., Shyy,W., Liu, H.: Near wake vortex dynamics of a hovering hawkmoth. Acta Mech. Sinica 25(1), 25–36(2009).