Aerodynamic CFD Study of Clark Y Airfoil

T. Prashanth[1,*], S. SampathKumar[1], P. Harshitha Reddy[1],
K.Chandra Shekar[1], Subhash Deo Hiwase[2]
[1.]Department of Mechanical Engineering, Vignan Institute of Technology and Science, Hyderabad, India.
[2]Axis-IT&T Limited, Hyderabad, India

Abstract— Airfoils have become an integral aspect of human flight as it has evolved over the last century. As the design of each airfoil determines many aspects of its use in the real world, all significant characteristics must be analyzed prior to implementation. The aerodynamic effects of pressure, drag and lift were evaluated by computational fluid dynamics method to determine the behavior of the Clark Y airfoil with an air velocity of 0.38 Mach in turbulent condition. In the present study pressure, velocity and turbulent kinetic energy distribution was recorded over the upper and lower surfaces of the airfoil derived by ANSYS, a computer simulation package with Fluent solver.

In the same respect, pressure measurements at various angles of attack were taken directly downstream of the airfoil to determine stall condition, and in turn the lift force on the Clark Y. In this analysis the maximum stall angle was calculated.

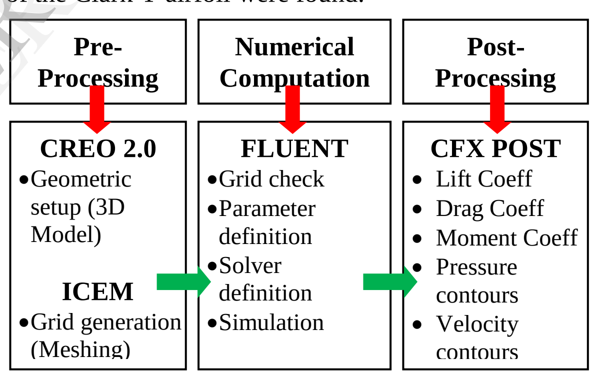
Keywords— ***Aerodynamic, Clark Y airfoil, CFD, Turbulent condition, stall angle***

I. INTRODUCTION

Waves follow our boat as we meander across the lake, and turbulent air currents follow our flight in a modern jet. Mathematicians and physicists believe that an explanation for and the prediction of both the breeze and the turbulence can be found through an understanding of solutions to the Navier-Stokes equations. Although these equations were written down in the 19th Century, our understanding of them remains minimal. The challenge is to make substantial progress toward a mathematical theory, which will unlock the secrets hidden in the Navier-Stokes equations. Based on the continuum hypothesis, which considers that the, physical properties characterizing the state of a fluid such as pressure, density, velocity, etc. vary continuously, equations describing the motion of fluids can be derived without regarding the behavior of individual molecules [1]. The complexity of finding solutions of Navier Stokes equations is so great that such a problem has been put in the list of the greatest unsolved problems [2]. So, one way of progress in the field is resorting to the use of numerical methods [3]. That is, CFD is basically a relative new field of science that includes all the steps involved in simulating a fluid flow on the computer. With the advent of microprocessors and chip technology in last decade there is an exponential increase in computing power available, which continues to compact down even as we speak [4]. The availability of such computing power has been aiding growth and advances in the commercial aerospace, automobile, scientific and other engineering across the shores and redefining the design arena. Computational fluid dynamics had been spear heading this science and technology progression by being cost effective, determining solutions for complex design challenges, incorporation of advanced CFD analysis in ANSYS codes and solvers for optimizing dynamic simulations demanded by the transport vehicles of present and preparing us for future [5]. Some studies were also carried out to find the CFD uncertainty [6-9]. This paper aimed to predict lift forces at various angles of attack and to determine the stall angle.

II. METHODOLOGY

The computational steps in this study consist of three stages as shown in Figure 1. This study began from preprocessing stage of geometry setup and grid generation. The geometry of the model was drawn using CREO 2.0.The grid was generated by ICEM. The second stage was computational simulation by FLUENT solver using Finite volume approach. Finally is the post-processing stage, where the aerodynamic characteristics of the Clark Y airfoil were found.



Finite volume methodology is used for the solution in this analysis (Figure 2). It is a method for representing and evaluating partial differential equations in the form of algebraic equations. Similar to the finite difference method or finite element method, values are calculated at discrete places on a meshed geometry. "Finite volume" refers to the small volume surrounding each node point on a mesh. In the finite volume method, volume integrals in a partial differential equation that contain a divergence term are converted to surface integrals, using the divergence theorem. These terms are then evaluated as fluxes at the surfaces of each finite volume. Because the flux entering a given volume is identical to that leaving the adjacent volume, these methods are conservative. Another advantage of the finite volume method is that it is easily formulated to allow for unstructured meshes.

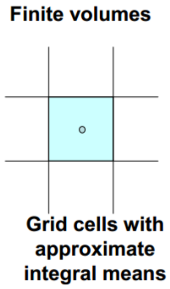


Figure 2. Finite volume methodology

In this study k- Epsilon two-equation turbulent model was used to determine the forces

Formulas for computing the turbulence model variables

Once an appropriate turbulence intensity and turbulence length scale or eddy viscosity ratio have to be estimated or measured (Generally default values in the simulation software are taken), the primitive turbulence model variables can be computed from the following formulas.

Modified turbulent viscosity:

The modified turbulent viscosity, $\tilde{\nu}$, can be computed using the following formulas:

From the turbulence intensity and length scale

$$\tilde{\nu} = \sqrt{\frac{3}{2}}\,(U\,I\,l)$$

Where,

U is the mean flow velocity,
I is the turbulence intensity and
L is the turbulent length scale.

Ideally, , but some solvers can have problem with that so $\tilde{\nu} <= \frac{\nu}{2}$ can be used. This is if the trip term is used to "start up" the model i.e. to initiate the solution. A convenient option is to set $\tilde{\nu} = 5\nu$ in the free stream. The model then provides fully turbulent results and any regions like boundary layers that contain shear become fully turbulent.

Turbulent energy

The turbulent energy, K, can be computed as:

$K = 0.5\,(U\,I)^2$

Where,

U is the mean flow velocity and
I is the turbulence intensity.

Dissipation rate:

The turbulent dissipation rate, ϵ, can be computed using the following formulas:

From the turbulence length scale

$$\epsilon = C_\mu^{\frac{3}{4}} \frac{k^{\frac{3}{2}}}{l}$$

K is the turbulent energy and
L is the turbulent length scale

C_u is a turbulence model constant which usually has a value of 0.09

III. Computational Method

The airfoil chosen for this experiment was the Clark Y, a general purpose airfoil used for its superb control at low Reynolds numbers. An additional feature of this airfoil is that its lower surface is parallel to its chord and this enables the use of an inclinometer to change the angle of attack directly. In this CFD analysis, Clark Y airfoil is characterized under medium-speed operating conditions. All other boundary conditions and the selected values of various parameters are mentioned below in the section 3.4.

A. Geometry

A 3D flow experiment was conducted by taking the airfoil geometry with a section of 10mm thickness as mentioned in the Figure 3 to measure reaction forces directly. The data obtained from the study was then analyzed in order to obtain values for the coefficient of lift. The stall angle was also calculated. These results were used to build a more rigorous analysis of the Clark Y airfoil's behavior under medium speed conditions.

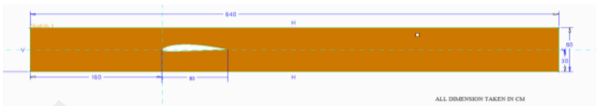


Figure 3. Geometry of the selected airfoil

B. Meshing

Meshing is the discretization or dividing the geometry into number of nodes and elements. The value change in the adjacent elements drastically vary near the solid object, therefore for the discretization to be continuous function the size of the element near the solid i.e., airfoil is put very small and unstructured mesh is obtained at that site.

Additional mesh details like 'Edge sizing' is also used in order to get the required mesh at the airfoil edges with the element size as 0.0004 mm as shown in Figure 4

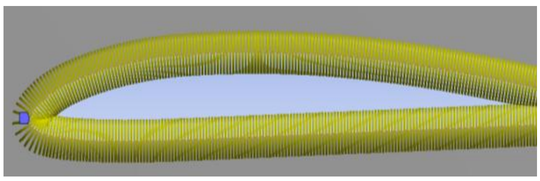


Figure 4. Edge Sizing function

The Result of the Generated mesh with the above parameters is shown in Figure 5 and Figure 6.

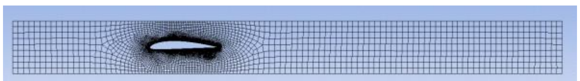


Figure 5. Mesh generated as a result of inputs

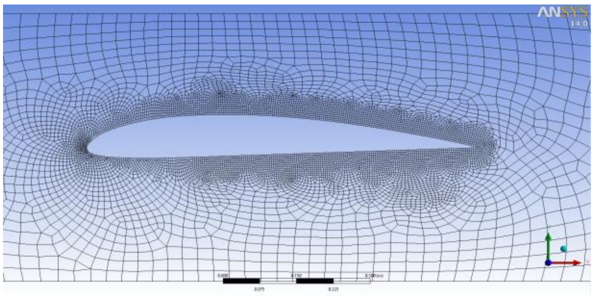


Figure 6. Unstructured mesh at the airfoil region

C. Grid Independence Test

Grid convergence is the term used to describe the improvement of results by using successively smaller cell sizes for the calculations. A calculation should approach the correct answer as the mesh becomes finer, hence the term grid convergence. The normal CFD technique is to start with a coarse mesh and gradually refine it until the changes observed in the results are smaller than a pre-defined acceptable error. There are two problems with this approach. Firstly, it can be quite difficult with other CFD software to obtain even a single coarse mesh result for some problems (particularly when time is pressing). Secondly refining a mesh by a factor of 2 can lead to a8-fold increase in problem size so even more time is needed. This is clearly unacceptable for a piece of software intended to be used as an engineering design tool operating to tight production deadlines. These and other issues have added greatly to the perception of CFD as an extremely difficult, time consuming and hence costly methodology.

There are several types of testing categories of which skewness is considered in this analysis. Skewness is a measure of the asymmetry of the probability distribution of a real-valued random variable about its mean. The value obtained is 0.73.

D. Solution Setup

In this step the boundaries are named and their boundary conditions are defined. All the parameters required for solving the fluid flow equations are defined in this step. The solver we are using for the analysis is Fluent. All the data given to it, is generated as input summary, which is as follows shown in Table 1,2 and 3.

Boundary Conditions	
Inlet	Velocity inlet of 130 m/s
Outlet	Pressure outlet 0 bar (absolute pressure)
Airfoil	Wall

Table 1. Boundary conditions

Discretization Scheme	
Pressure	Standard
Momentum	Second order upwind
Turbulent kinetic energy	First order upwind
Turbulent dissipation rate	First order upwind

Table 2. Discretization Scheme

Relaxation Factors	
Pressure	0.3
Density	1
Body forces	1
Momentum	0.7
Turbulent kinetic Energy	0.8
Turbulent dissipation rate	0.8
Turbulent viscosity	1

Table 3. Relaxation factors

E. Scope and Limitations

At higher angles of attack, a large separation region in the downstream of the airfoil is observed. This separation region makes the flow unsteady. Because the time dependent features of the flow field are neglected in this simulation, the flow predictions for higher angles of attack will not be accurate.

This exercise is aimed for resolving high-speed flows only. It is not possible to solve the flow system with laminar conditions. Difficulty in obtaining convergence or poor accuracy may result if input values are used outside the upper and lower limits suggested in the problem overview [10].

IV. Results and Discussion

Static Pressure (Pa), Velocity magnitude (m/s) & turbulent kinetic energy (m^2/s^2) respectively is plotted in the contour map for the Clark Y airfoil at angle of attacks of 0 to 30 degrees at an interval of 2 degrees and at free stream velocity of 130 m/s.

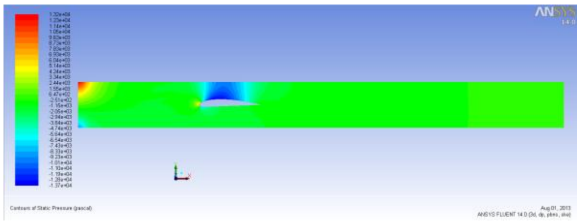


Figure 7. Static pressure contours at 22^o angle of attack

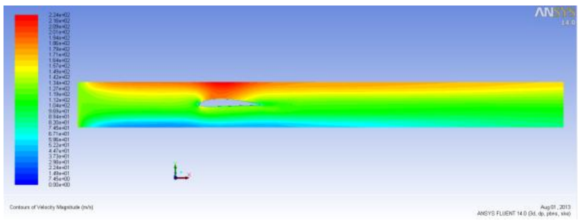


Figure 8. Velocity contours at 22^o angle of attack

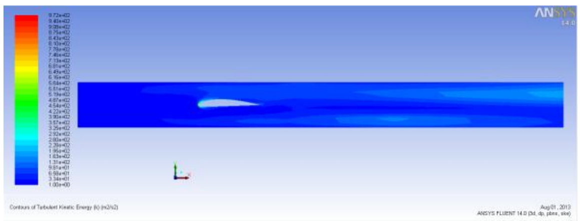


Figure 9. Turbulence contours at 22^o angle of attack

Clark Y airfoil is generally known for its high stability at low Reynolds Number i.e. is very low speed. This analysis is carried out to find out the maximum stall angle at higher Reynolds Number and study the aerodynamic behavior. The Result for the given Reynolds number 5.2e+06 is about 22 Degrees. The static pressure contour, Velocity contour and turbulent kinetic energy contour are shown in Figure 7,8 and 9 respectively.

The study was successful at plotting the graph against Coefficient of lift and angle of attack. It is observed that, as the angle of attack increases, the static pressure shifts over the airfoil as the airfoil stalls. It is also clear that as the angle of attack increases the flow becomes unsteady; the reason for this unsteady behavior of flow is due to the drastic increase in the turbulent kinetic energy and dissipation energy. Hence, altering the pressure distribution over the upper surface of the airfoil. The values of lift forces and their coefficients at different angles of attack ranging from 0^{O} to 30^{O} with a 2-degree interval are given in Figure 10.

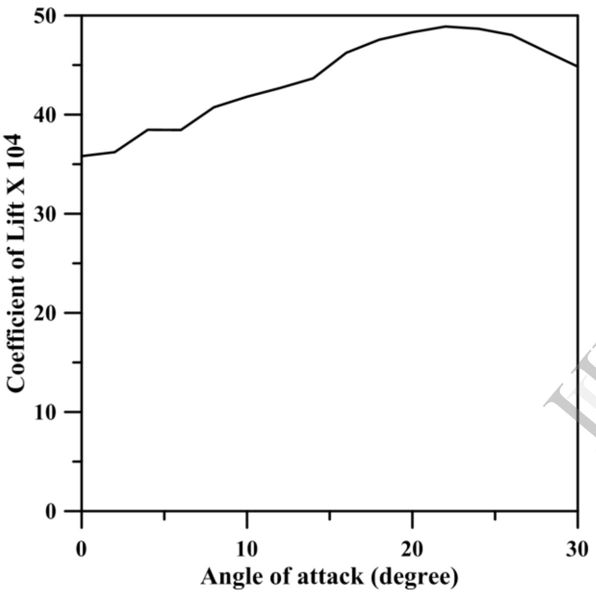


Figure 10. Coefficient of lift Vs angle of attack

From Figure 10 it is clear that coefficient of lift increases with increasing angle of attack till 22^{o} after which the trend is changed. The lift generated at this point is maximum with the proportional amount of drag.

V. CONCLUSION

Clark Y airfoil at the given conditions of input velocity 130 m/s results in maximum lifting force at 22^{O} angle of attack. The lift force generated at the maximum angle of attack for the geometric section is about 50.58N.

ACKNOWLEDGEMENT

The authors gratefully acknowledge Dr. M. Venkata Ramana, Prof. N. Leela Prasad, and Prof. S. Venugopal Rao for their constant encouragement during this work.

REFERENCES

[1] Batchelor.G.K. *An introduction to fluid dynamics*, Cambridge Univ. Press, 1967

[2] Fletcher.C.A.J.(), *Computational techniques for fluid dynamics*, 2nded.,vol.1&2,Springer-Verlag, 1991

[3] Roache P. J., *Verification and validation in compu- tational scienсe and engineering[M]*, Hermosa Publishers, Albuquerque, New Mexico, USA:, 1998.

[4] Samuel H. Fuller and Lynette I. Millett, *The future of computing performance*, The National Academies Press, Washington, D.C

[5] Zhen-qiu YAO, Hong-cui SHEN, Hui GAO, “A new methodology for the CFD uncertainty analysis”, *Journal of Hydrodynamics*, Ser.B, Volume25,Issue 1, February 2013, pp 131-147 [6] Campana E F., Peri D. and Tahara Y. et al, “Comparison and validation of CFD based local optimi- zation methods of surface combatant bow[C]”. *25th Symposium on Naval Hydrodynamics*, St. John’s Newfoundland and Labrador, Canada, 2004.

[7] Zhu De-xiang, Zhang Zhi-rong and WU Cheng- sheng et al, “Uncertainty analysis in ship CFD and the primary application of ITTC procedures[J]”, *Journal of Hydrodynamics*, Ser. A, 2007, pp 363-370.

[8] ZHANG Nan, SHEN Hong-cui and YAO Hui-zhi, “Un- certainty analysis in CFD for resistance and flow field[J]”. *Journal of Ship Mechanics*, 2008, pp 211- 224.

[9] Yao Zhen-qiu, Yang Chun-lei and Gao Hui, “Nume rical simulation of turbulent flow around a submarine and its uncertainty analysis[J]”, *Journal of Jiangsu,* University of Science and Technology, 2009, pp 95-98.

[10] Yang Ren-you, Shen Hong-cui and Yao Hui-zhi. “Numerical simulation on self-propulsion test of the sub- marine with guide vanes and calculation for self-propulsion factors[J]”, *Journal of Ship Mechanics*, 2005, pp 1-40.