Computational Fluid Dynamics Study Of Fluid Flow And Aerodynamic Forces On An Airfoil

S.Kandwal[1] , Dr. S. Singh[2]

[1]M. Tech Scholar, [2]Associate Professor

Department of Mechanical Engineering,
Bipin Tripathi Kumaon Institute of Technology, Dwarahat, Almora, Uttarakhand (India) 263653

Abstract

This paper presents computational investigation of invicid flow over an airfoil. The drag and lift forces can be determined through experiments using wind tunnel testing, in which the design model has to be placed in the test section. The experimental data is taken from Theory of Wing Sections by Abbott et al., This work presents a computational method to deduce the lift and drag properties, which can reduce the dependency on wind tunnel testing. The study is done on air flow over a two-dimensional NACA 4412 Airfoil using ANSYS FLUENT (version 12.0.16), to obtain the surface pressure distribution, from which drag and lift were calculated using integral equations of pressure over finite surface areas. In addition the drag and lift coefficients were also determined. The fluid used for this purpose is air. The CFD simulation results show close agreement with those of the experiments, thus suggesting a reliable alternative to experimental method in determining drag and lift.

Keywords: Flow over airfoil; pressure coefficient; CFD analysis; GAMBIT

1. Introduction

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, windmill blades, compressor and turbine blades in a jet engine, and hydrofoils are example airfoils. The basic geometry of an airfoil is shown in Figure 1. Understanding motion of air (often called a flow field) around an object enables the calculation of forces and moments acting on the object.
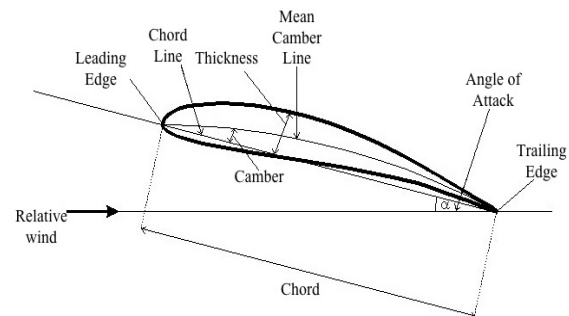

Figure 1: Diagram of airfoil's geometry

Aerodynamics allows the definition and solution of equations for the conservation of mass, momentum, and energy in air. The use of aerodynamics through mathematical analysis, empirical approximations, wind tunnel experimentation, and computer simulations form the scientific basis for heavier than air flight and a number of other technologies. Aerodynamic problems can be classified according to the flow environment. External aerodynamics is the study of flow around solid objects of external aerodynamics. Internal aerodynamics is the study of flow through passages in solid objects. For instance, internal aerodynamics encompasses the study of the airflow through a jet engine or through an air conditioning pipe. Some problems may encounter only very small viscous effects on the solution, in which case viscosity can be considered to be negligible. The approximations to these problems are called inviscid flows. Flows for which viscosity cannot be neglected are called viscous flow.

Alex et al., (2010) perform a Wind tunnel testing on NACA 3314, NACA 8321, NACA 1209, NACA 6217, NACA 0014, and NACA 5417 airfoils. The experimentation allowed a comparison of flight characteristics between the airfoils, in which each generated an expected range of lift and drag forces. *G.V. et al., (1983)* presented new linearized theory for pitching moment characteristics of two-element airfoil and for a correction method for a new type of slotted wall wind tunnel. Comparisons with exact numerical calculations and with experimental data show good agreement with prediction of the linearized theory. *Nicholas et al., (2002)* he design problem selected for this project is the design of a low Reynolds number (100,000 to 1,000,000) airfoil to be used on Uninhabited Aerial Vehicles or UAVs.

2. Analytical Solution

Aerodynamic drag is caused by dynamic interaction between a body surface and the fluid which flows over it. Two major terms which govern the aerodynamic drag and lift are the normal stress and wall shear stress. Pressure distribution dominates the normal stresses acting on the body surface, while surface roughness contributes the wall shear stress. The equations for calculating lift and drag are very similar. The lift that an airfoil generates depends on the density of the air, the velocity of the airflow, the viscosity and compressibility of the air, the surface area of the airfoil, the shape of the airfoil, and the angle of the airfoil's angle of attack.

However, dependence on the airfoil's shape, the angle of attack, air viscosity and compressibility are very complex. Thus, they are characterized by a single variable in the lift equation, called the lift coefficient. Therefore, the lift equation is given by

$$L = \frac{1}{2}\rho U^2 S C_L \tag{1}$$

Where L is the lifting force, ρ is the density of air, U is the relative velocity of the airflow, S is the area of the airfoil as viewed from an overhead perspective, and C_L is the lift coefficient. As with lift, the drag of an airfoil depends on the density of the air, the velocity of the airflow, the viscosity and compressibility of the air, the surface area of the airfoil, the shape of the airfoil, and the angle of attack.

The drag coefficient is generally found through testing in a wind tunnel, where the drag can be measured, and the drag coefficient is calculated by rearranging the drag equation

$$D = \frac{1}{2}\rho U^2 A C_D \tag{2}$$

In the drag equation, D is the drag force, ρ is the density of the air, U is the velocity of the air, A is a reference area, and C_D is the drag coefficient.

Pressure coefficient:

The pressure coefficient can be expressed by

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho U^2} \tag{3}$$

Lift coefficient:

If the lift coefficient for a wing at a specified angle of attack is known then the lift produced for specific flow conditions can be determined using the following equation.

The lift of the airfoil can also be expressed as

$$\text{Lift} = F_L = \iint_{surface} dF_L = \left[\int_0^L dF \cos \theta\right]_{top} + \left[\int_0^L dF \cos \theta\right]_{bottom} \tag{4}$$

or simplicity this span is considered uniform, because of this, integration is only necessary over L given

$$\int_0^L dF \cos \theta = \int_0^L ps \; dl \cos \theta = \int_0^L (p - p_\infty) s \; dl \; \cos \theta \tag{5}$$

It is known that $\cos \theta = \frac{dx}{dl}$ yielding,

$$\int_0^L (p - p_\infty) s \; dl \; \frac{dx}{dl} = s \int_0^L (p - p_\infty) \; dx \tag{6}$$

With the distance along the chord defined as
dx=dc (cos α),

$$S\int_0^L (p - p_\infty) \cos \alpha \; dc = F_L \tag{7}$$

It is known from equation (1) that,

$$C_L = \frac{F_L}{\frac{1}{2}\rho U^2 A} \qquad (8)$$

Drag coefficient:

In fluid dynamics the drag coefficient commonly denoted as C_D is a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment such as air or water. It is used in the drag equation where a lower drag coefficient indicates the object will have less aerodynamic or or drag. The drag coefficient is always associated with a particular surface area. The drag coefficient of any object comprises the effects of the two basic contributors to fluid dynamics drag: skin friction and from drag. The drag coefficient of a lifting airfoil or hydrofoil also includes the effects of lift induced drag. The drag coefficient of a complete structure such as an aircraft also includes the effects of interference drag. The overall drag coefficient defined in the usual manner is

$$C_D = \frac{F_D}{\frac{1}{2}\rho U^2 A} \qquad (9)$$

The reference area depends on what type of drag coefficient is being measured. For automobiles and many other objects, the reference area is the projected frontal area of the vehicle. This may not necessarily be the cross sectional area of the vehicle, depending on where the cross section is taken. For example, for a sphere $A = \pi r^2$ (this is not the surface area $A = \pi r^2$). For airfoils the reference area is the plan form area.

3. Modeling and Simulation
Description of the Physical Model:
A schematic of the geometric model of the Airfoil used in the study is shown in figure.
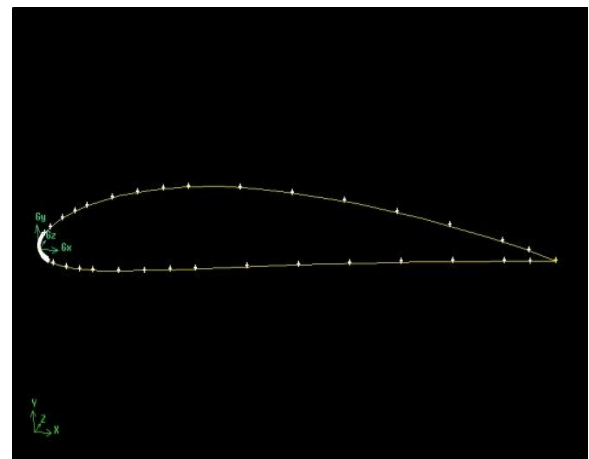

Figure 2: geometry of Airfoil created in GAMBIT

To conduct numerical simulation, the computational domain as shown in figure, is meshed with control volume built around each grid using GAMBIT (version 2.3.16) which is the preprocessor for ANSYS FLUENT (version 12.0.16). The mesh for airfoil simulation is an unstructured type

consisting of 1, 12150 cells and 12405 nodes. The grid independence test was performed to the quality of mesh for solution accuracy. air flow over a two-dimensional (2-D) Airfoil object was considered, with the Assumption that the fluid movement in the third component (z-direction) was negligible. The fluid properties were set to be similar to that used in the experiment reported in.

Boundary Conditions:

The flow is consider invicid, two dimensional and steady with constant thermodynamic properties with the inlet temperature 288.17 K. the working fluid is air with Mach no 0.15. The parameters are associated with practical Airfoil operating conditions; the air flow has an air temperature of 288.17 K and Velocity of 50 m/s. The properties of air at temperature 288.17 K and the dimension parameters of airfoil are shown in **Table 1.**

Properties of air at 288.17 K	Dimension parameters of Airfoil
Density: 1.2250 Kg/m^3	Cord: 100 mm
Kinematics Viscosity: 1.4607×10^{-5}	Area: 100 m^2
Specific heat: 1.4 kj/kg K	Mack No: 0.15
-	Length of trailing edge: 0.02c
-	Angle of attack: 2°

The mesh is exported to ANSYS FLUENT 12.0 along with the physical properties and the initial conditions specified. The material properties and the initial conditions are read through the case file. Instructions for the solver are provided through a journal file. When the solution is converged or the specified number of iterations is met, FLUENT exports data to a mesh file and to XY plot files.

4. **Results and Discussion**

Computational Fluid Dynamics (CFD) has shown to be adequate for predicting the pressure forces on Airfoil. The CFD results for a certain case are compared to experimental results and then, if found good, the numerical results of other similar cases are considered as accepted. The coefficient of Drag and coefficient of Lift for regions were also compared with experimental results and are shown in **Table 2.**

Coefficient of forces	Experiment (Albert,pg. 488)	Numerical (Present study)
C_L	0.649	0.654
C_D	0.007	0.001

Result shows the value of lift coefficient calculated computationally is 0.654 while

the experimental value is 0.649 for invicid flow over the airfoil.

And the value of drag coefficient is 0.001 calculate computationally and 0.007 experimentally. We used inviscid case for our model, so we are expecting a C_D of zero.
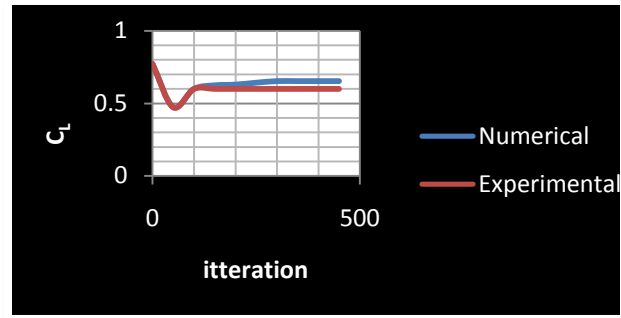

Figure 4: comparison of coefficient of limit in Experimental and numerical value (in MS-Excel)

Figure 4 shows that the simulation results correlate well with the theories related to lift and drag, thus providing reliable calculations of lift and drag coefficients using the presented method.
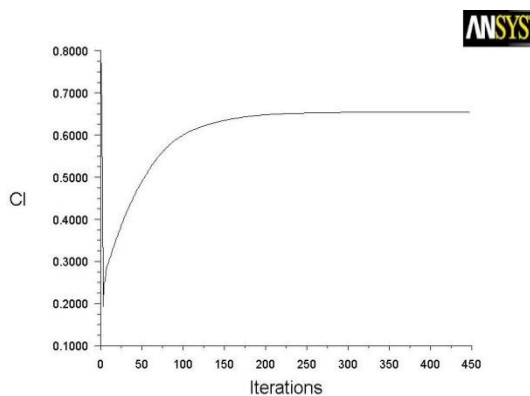
Figure 5: Lift Coefficient

The simulation results were analyzed in various stages. The post-processing features in ANSYS fluent 12.0 are able to provide several types of contour plot, such as pressure and velocity plots.
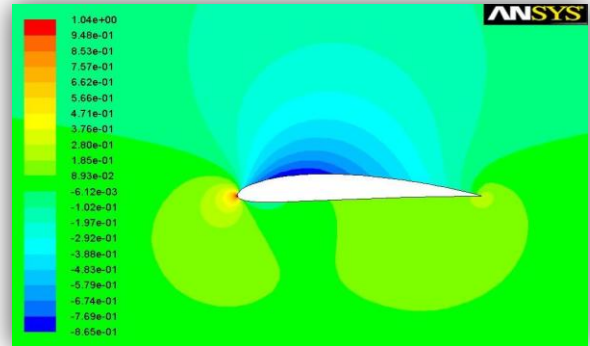

Figure 6: Contour plot of Pressure coefficient

Figure 6 shows the pressure coefficient contour plot in the flow regime for M of 0.15. From the contour of pressure coefficient; it is show that there is a region of high pressure at the leading edge (stagnation point) and region of low pressure on the upper surface of airfoil.
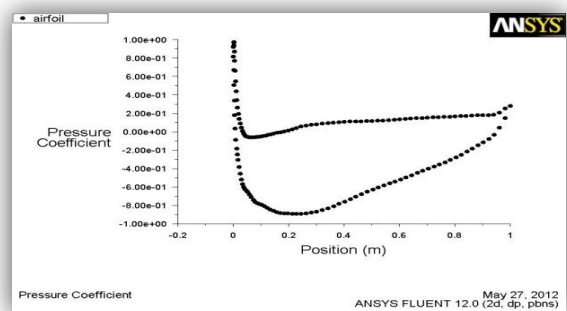

Figure 7: Variation of pressure coefficient for air along airfoil at different position

Figure 8 shows velocity magnitude contour plots generated by the post-processing feature for better understanding of velocity profiles an important aerodynamic property of interest in the study of drag is the wake region, which can be determined from the velocity vector plot
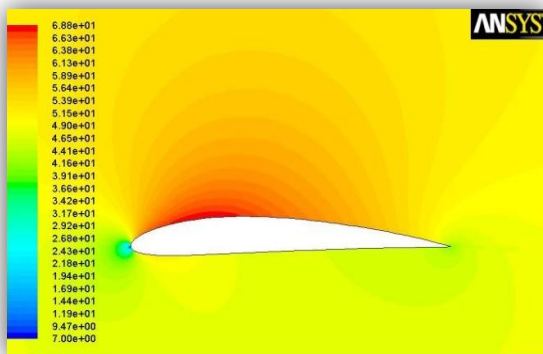

Figure 8: Contours of velocity magnitude

On the leading edge, we see a stagnation point where the velocity of the flow is nearly zero. The fluid accelerates on the upper surface as can be seen from the change in colors of the contour.
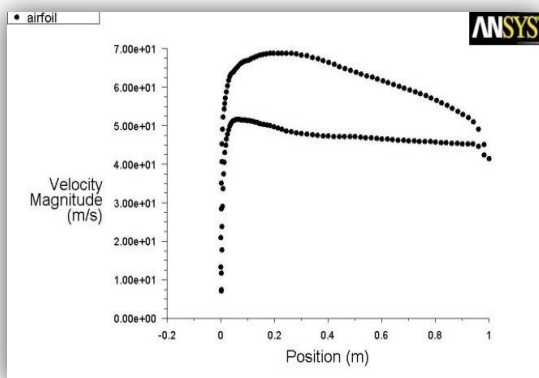

Figure 9: Variation of velocity magnitude for air along airfoil at different position

Comparison of the numerically derived results with that of the experiments shows a good correlation. Hence the proposed analysis method has demonstrated a workable alternative to obtain drag and lift coefficients by manipulating the results from ANSYS simulation. However further investigations are suggested in order to reduce the differences in the results at certain conditions, and to enable calculations of friction related drag and lift.

5. Conclusions

Based on the CFD analysis of the flow over airfoil the following conclusions can be drawn:

1. Pressure coefficient is maximum at the point of attack and minimum the upper surface of airfoil.

2. The velocity of the upper surface is faster than the velocity on the lower surface.

3. Computed lift and drag forces were found in close agreement with the experimental values.

6. Acknowledgements

me in all the technical and non-technical issues during the production of this work. His encouragement and efforts led this report to successful completion in a timely fashion**.**

7. References

[1] Anderson, John D.,
"Fundamentals of Aerodynamics" 2nd Edition, McGraw-Hill, New
York, 1991.

[2] Albert Edward Von Doenhoff
"Theory of Wing Sections" pg. 488

[3] Alex Sullivan Cleveland, F. A.,
"Aerodynamic forces acting on an airfoil"

[4] Brenda M. Kulfan and John E. Bussoletti "Fundamental" Parametric Geometry Representations for Aircraft Component Shapes. AIAA—2006-6948

[5] Ghias, R., Mittal, R., and Lund, T.,
"A Non-Body Conformal Grid Method for Simulation of Compressible Flows with Complex Immersed Boundaries," AIAA Paper 2004-0080, 2004.

[6] G.V.Parkinson and G.D.watt
"Some new applications of linearized airfoil theory" (1983)

[7] Ira Herbert Abbott, Albert Edward Von Doenhoff "Theory of Wing Sections"

[8]Nicholas K. Borer
"Design and Analysis of Low Reynolds Number Airfoils" Dec. 2002