Calculation of Aerodynamic Characteristics of NACA 2415, 23012, 23015 Airfoils Using Computational Fluid Dynamics (CFD)

Himanshu Parashar

***Abstract*— A method of solving the flow over airfoils of National Advisory Committee for Aeronautics (NACA) series - 2415, 23012 & 23015 is proposed using the assistance of commercial CFD package's such as Gambit and Fluent. The flow was obtained by solving the steady-state governing equations of continuity, momentum, and energy conservation combined with standard k-ε turbulence model. NACA airfoils with angle of attack from -15 to +15 degrees with an interval of 5 degrees are analyzed. Calculations were made for constant air velocity altering only the angle of attack for each airfoil.**

Various meshes were generated to check grid dependency. The aerodynamic characteristics of the airfoils calculated in Fluent software by the first-order quantity with two dimensional double precision solver. It is shown that the effects on the aerodynamic characteristics such as lift and drag coefficients of airfoils are associated with their shapes. The NACA 4 digit and 5 digit airfoils behaves differently in same flow fields which in turn change the lift, drag and moment of airfoils.

***Index Terms*— Aerodynamics characteristics, NACA airfoils, Lift and Drag, Turbulence model**

I. INTRODUCTION

The flow around airfoils or cascade of airfoils is known to be highly three dimensional, unsteady, & turbulent, making the modeling of the flow very demanding in terms of computational resources. Some of the important factors to be considered in the analysis of the flow are the adverse pressure gradient and the presence of high turbulence stress. When simulating the flow over airfoils, transition from laminar to turbulent flow plays an important role in determining the flow features and in quantifying the airfoil performance such as lift and drag. Hence, the proper modeling of transition, including both the onset and extent of transition will definitely lead to a more accurate drag prediction. If a number of flow situations are to be considered, a number of physical geometries are to be created and then tested. This increases time and expense of testing. The development of high speed digital computing during the last few decades has brought the great impact on the way a physical situation is modeled and analyzed. Speed of modern computing and principles from the science of fluid mechanics, heat transfer, and combustion are applied to engineering design practice.

Numerical methods have emerged as a powerful method and have overcome the restrictions in both experimental and analytical methods. They solve the discretization of the governing mathematical equations in a way such that the numerical solutions can be obtained. This approach forms the core of Computational Fluid Dynamics.

II. NUMERICAL METHOD AND TURBULENCE MODELING

In the present study the numerical simulation was carried out using the finite volume based CFD code FLUENT for solving the Reynolds-averaged Navier–Stokes (RANS) equations. The mass conservation equation, the full Navier stokes equation, and energy equations are the governing equations used in this study. The equation for conservation of mass or continuity equation can be written as follows:

Continuity Equation

Equation (1) is the general form of the mass conservation equation and is valid for incompressible as well as compressible flows.

$$\frac{D\rho}{Dt} + \rho \frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

Momentum Equation

Conservation of momentum is described by Equation (2).

$$\underbrace{\rho \frac{\partial U_j}{\partial t}}_{I} + \underbrace{\rho U_i \frac{\partial U_j}{\partial x_i}}_{II} = -\underbrace{\frac{\partial P}{\partial x_j}}_{III} - \underbrace{\frac{\partial \tau_{ij}}{\partial x_i}}_{IV} + \underbrace{\rho g_j}_{V} \tag{2}$$

Where

$$\tau_{ij} = -\mu \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) + \frac{2}{3} \delta_{ij} \mu \frac{\partial U_k}{\partial x_k}$$

I: Local change with time

Manuscript received March 16, 2015.
***First Author name**, Himanshu Parashar, Student of International Masters in Turbulence, École Centrale de Lille, France*

ISSN: 2278 – 7798

II: Momentum convection
III: Surface force
IV: Molecular-dependent momentum exchange (diffusion)
V: Volume force

Energy Equation

Conservation of energy is described by Equation (3).

$$\underbrace{\rho c_\mu \frac{\partial T}{\partial t}}_{I} + \underbrace{\rho c_\mu U_i \frac{\partial T}{\partial x_i}}_{II} = \underbrace{-P \frac{\partial U_i}{\partial x_i}}_{III} + \underbrace{\lambda \frac{\partial^2 T}{\partial x_i^2}}_{IV} - \underbrace{\tau_{ij} \frac{\partial U_j}{\partial x_i}}_{V} \quad (3)$$

Where

I : Local energy change with time
II: Convective term
III: Pressure work
IV: Heat flux (diffusion)
V: Irreversible transfer of mechanical energy into heat

The second order upwind scheme has been used to solve the flow variables. The pressure based Navier-Stokes solver is used for the analysis of the problem. The simplest "complete models" of turbulence are two equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard k-ε model falls within this class of turbulence model and has been used in present case.

III. PHYSICAL MODEL AND BOUNDARY CONDITIONS

The 2-Dimensional airfoil geometries were created in CAD (CATIA) environment using the control point of the camber profile and meshing was done in Gambit software. A Pitch - Chord ratio of 0.5 C was maintained in all cases.

The airfoil is assumed to have a chord length of 1.0 meters (all dimensions are in standard SI system of units), with the front-most part of the airfoil at (0, 0, and 0) and the trailing edge at (1, 0, 0).

The computational domain extends far upstream of the airfoil where the boundary condition are defined as velocity inlet and outlet was defined as pressure outlet. Inlet velocity was set to 50 m/s and outlet pressure was set to atmospheric pressure. The periodic boundary conditions were defined on lower and upper side of the domain. On all the solid boundaries, the conditions invoked are the No-slip condition and heat flux normal to the wall, q = 0. Computational domain is shown in fig 1.

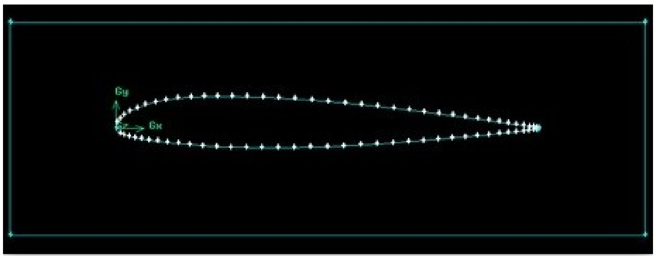

Fig. 1 Computatonal Domain

IV. GRID GENERATION

In order to adequately resolve the boundary layer along the airfoil wall, grid points are clustered near the wall. Far away from walls, where the flow does not have large velocity gradients, the grid points are kept a bit apart. A hybrid grid was used in this problem. An unstructured quad-pave mesh was generated over airfoil. Grid dependency was checked and found there was no change in result when total no of cells were around 15,000. The governing equations are discretized and solved at these grid points.

V. RESULTS

The numerical simulations are performed and results are reported for NACA 2415, 23012, 23015 airfoils. The angle of attack is in degrees for all the figures.

Fig. 2-4 shows the variation of coefficient of lift with angle of attack. It can be seen that C_l varies linearly with α over a large range of angle of attack. The lift is showing a negative trend as α is going below 0 degrees and a upward trend as angle of attack is increased above 0 degrees for all three airfoils.

When α = 0, there is still a positive value of C_l, that is, there is still some lift even when the airfoil is at zero angle of attack to the flow. This is due to positive camber of the airfoil. NACA 2415 airfoil gives the maximum lift compared to the NACA 23012 and 23015 airfoils irrespective of α. NACA 23012 and 23015 airfoil shows more or less same characteristics.

ISSN: 2278 – 7798

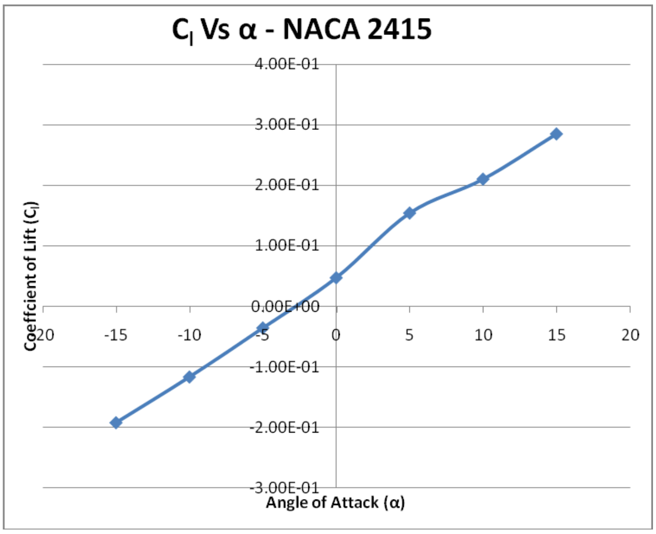

Fig. 2 C_l Vs α Plot for NACA 2415

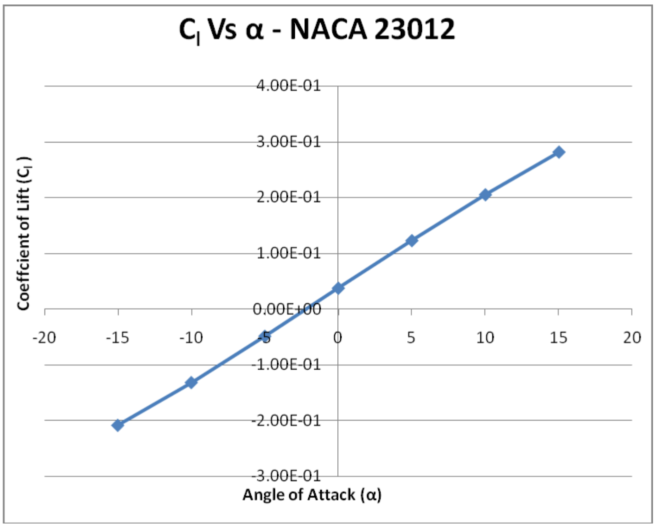

Fig. 3 C_l Vs α Plot for NACA 23012

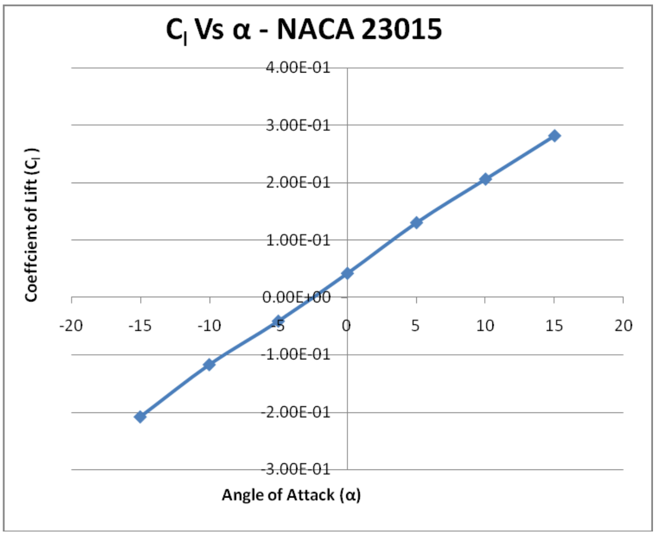

Fig. 4 C_l Vs α Plot for NACA 23015

Lift and drag varies with the angle of attack. In fact, drag is the price for generating the lift. Thus, although it is desirable to obtain as much lift as possible, this cannot be done without increasing the drag. It is therefore necessary to find the best compromise.

Fig. 5 - 7 represents the plot of coefficient of drag and their corresponding angle of attack. The maximum or peak drag is encountered when α= 0. Drag diminishes as 'α' increases or decreases. NACA 23015 and 2415 airfoil have same drag when $\alpha = 0$.

NACA 23012 airfoil have least drag at extreme values of angle of attack than that of the NACA 2415 and 23015.

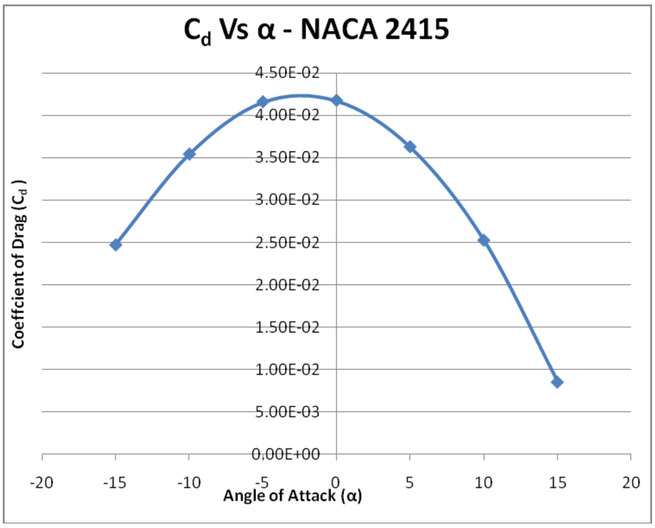

Fig. 5 C_d Vs α Plot for NACA 2415

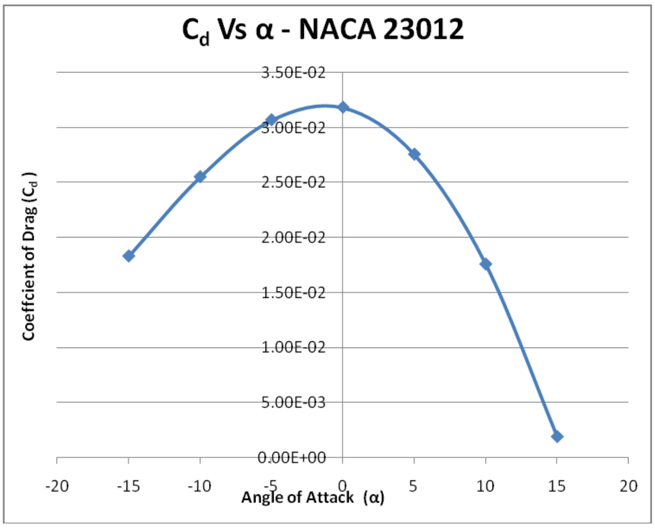

Fig. 6 C_d Vs α Plot for NACA 23012

ISSN: 2278 – 7798

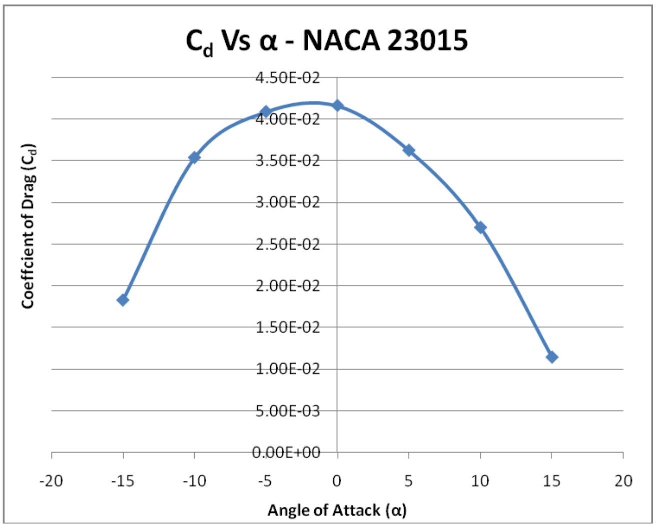

Fig. 7 C_d Vs α Plot for NACA 23015

The "drag polar" specifies the drag coefficient C_d for a given lift coefficient C_l(and vice versa). This is often the most important part of the results and can be used to find the best climb or sink rate as well as the optimum glide angle ideally possible with the airfoil. The reason for using the drag polar format is that when evaluating the aerodynamic performance of an airfoil, the α values are not really relevant. All that matters is the drag and how it compares to lift. The drag polar format compares these directly, and hence summarizes the most important features of the airfoil's drag characteristics in one plot.

The drag polar curve is shown in fig. 8-10 for NACA airfoils. These two parameters are of paramount importance since they are used to calculate aerodynamic efficiency. NACA 23012 airfoil has least drag compare to NACA 2415 and 23015 when $C_l = 0$. The NACA 23012 has maximum C_l at the least expense of mimum drag.Hence, its much better than rest two airfoils.

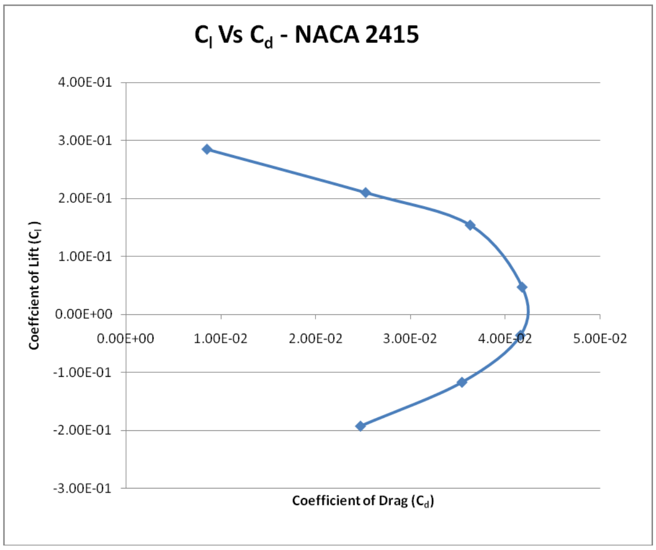

Fig. 8 C_l Vs C_d Plot for NACA 2415

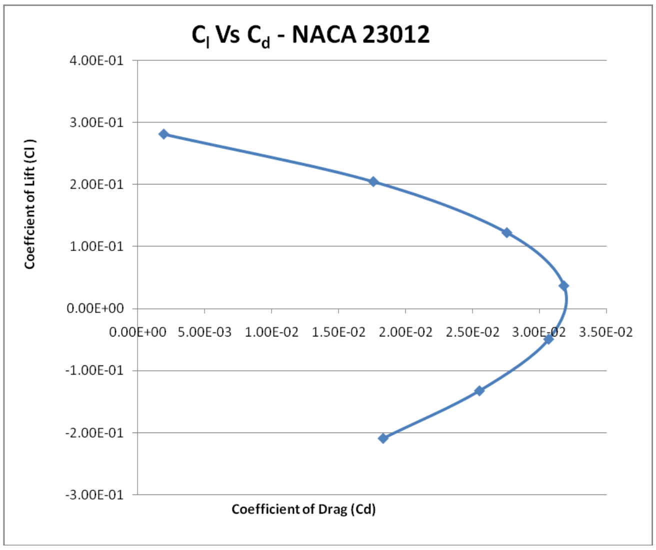

Fig. 9 C_l Vs C_d Plot for NACA 23012

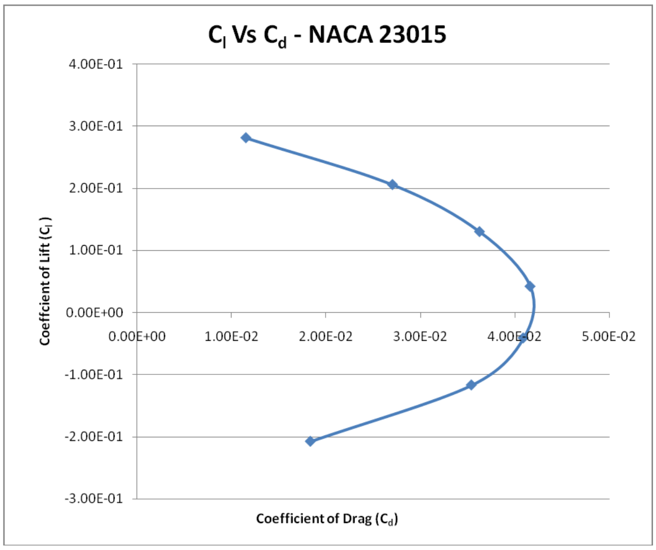

Fig. 10 C_l Vs C_d Plot for NACA 23015

The lift-drag ratio reaches its maximum at 0 degrees angle of attack, meaning that at this angle we obtain the most lift for the least amount of drag.

VI. CONCLUSION

Numerical simulations have been carried out for NACA 2415, 23012 & 23015 airfoil series. The simulation captures all the important features of the flow. A special care was taken to build good quality mesh with grid clustering to capture flow separation and adverse pressure gradient as much closely as possible. At a fixed freestream velocity with varying angle of attack ranging from -15 to +15 degrees, two main aerodynamic parameters C_l and C_d are calculated and plotted for all three NACA airfoils. A fair agreement is achieved between the available theory from Abbott et al and numerical simulation results for all the NACA airfoil.

ISSN: 2278 – 7798

ACKNOWLEDGMENT

I am greatly indebted to my esteemed institution Malla Reddy College of Engineering and Technology, Hyderabad, India. I extend my gratitude for the assistance provided by department faculty. The privilege of working on this project using CAD/CFD lab facilities remains memorable.

REFERENCES

[1] Abbott IH, Von Doenhoff AE (1959). Theory of Wing Sections. Dover Publishing, New York.
[2] Bacha WA, Ghaly WS (2006). Drag Prediction in Transitional Flow over Two-Dimensional Airfoils, Proceedings of the 44th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV.
[3] Badran O (2008). Formulation of Two-Equation Turbulence Models for Turbulent Flow over a NACA 4412 Airfoil at Angle of Attack 15 Degree, 6th International Colloquium on Bluff Bodies Aerodynamics and Applications, Milano, 20-24 July.
[4] Fluent Inc. (2006), Fluent 6.3 User's Guide.
[5] Fluent Inc.(2007), Gambit 2.4 User's Guide
[6] Johansen J (1997). Prediction of Laminar/Turbulent Transition in Airfoil Flows. Risø National Laboratory, Roskilde, Denmark.
[7] Menter FR (1994). Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA J., 32: 1598-1605.
[8] McCroskey WJ (1987). A Critical Assessment of Wind Tunnel Results for the NACA 0012 Airfoil. U.S. Army Aviation Research and Technology Activity, Nasa Technical Memorandum, 42: 285-330.

1) **First Author**: Himanshu Parashar has done Aeronautical Engineering from Malla Reddy College of Engineering and Technology, Hyderabad, India in the year 2010. He has worked as a Sr. CFD-Engineer in CSM Software Pvt. Ltd., Bangalore , India from 2010-2014. Presently he is pursuing Masters in Turbulence from École Centrale de Lille, France.

ISSN: 2278 – 7798