ICMIEE-PI-140416

Numerical Analysis and Comparison on Aerodynamics Characteristics of NACA-0012 & NACA-4412

G.M Hasan Shahariar [1,] *Mohammad Rashedul Hasan*[2], Mohammad Mashud[3]
[1,2,3]Department of Mechanical Engineering,
Khulna University of Engineering & Technology, Khulna-9203, BANGLADESH

ABSTRACT

The numerical analysis of the two dimensional subsonic flow over a NACA 0012 & NACA 4412 airfoil at various angles of attack which is operating at a Reynolds number of 3×10^6 is presented. A commercial computational fluid dynamic (CFD) code ANSYS FLUENT based on finite volume technique is used for the calculation of aerodynamics performance. The two dimensional model of the airfoil and the mesh is created through ANSYS Meshing which is run in Fluent for numerical iterate solution. The steady-state governing equations of Reynolds averaged Navier-Stokes is calculated for analyzing the characteristics of two-dimensional airfoils and the realizable k-epsilon model with Enhanced wall treatment is adopted for the turbulence closure. The aim of the work is to show the behavior of the airfoil at these conditions and to compare the aerodynamics characteristics between NACA 0012 & NACA 4412 such as lift co-efficient, drag co-efficient and surface pressure distribution over the airfoil surface for a specific angle of attack. Calculations were done for constant air velocity altering only the angle of attack for every airfoil model tested. This analysis can be used for the wing design and other aerodynamic modeling corresponds to these airfoil.

Keywords: *airfoil; cfd; rans; lift co-efficient; drag co-efficient; NACA 0012; NACA 4412*

1. Introduction

The rapid evolution of computational fluid dynamics (CFD) has been driven by the need for faster and more accurate methods for the calculations of flow fields around configurations of technical interest. In the past decade, CFD was the method of choice in the design of many aerospace, automotive and industrial components and processes in which fluid or gas flows play a major role. In the fluid dynamics, there are many commercial CFD packages available for modeling flow in or around objects. The computer simulations show features and details that are difficult, expensive or impossible to measure or visualize experimentally. 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. [1]

For numerical simulation the first step is to construct the model of the geometry and flow domain. The body about which flow is to be analyzed requires modeling. This generally involves modeling the geometry with a CAD software package. The second step is to establish the boundary and initial conditions. Since a finite flow domain is specified, physical conditions are required on the boundaries of the flow domain. The simulation generally starts from an initial solution and uses an iterative method to reach a final flow field solution. The third step is the generation of grid i.e meshing. The flow domain is discretized into a grid. Currently all cases involve multi-block, structured grids. The grid should exhibit some minimal grid quality as defined by measures of orthogonality (especially at the boundaries), relative grid spacing (15% to 20% stretching is considered a maximum value), grid skewness, etc. The next step is to establish the simulation strategy and set up input parameters. The strategy for performing the simulation involves determining such things as the use of space-marching or time-marching, the choice of turbulence or chemistry model, and the choice of algorithms. The NACA four-digit wing sections define the profile by:

- First digit describing maximum camber as percentage of the chord.
- Second digit describing the distance of maximum camber from the airfoil leading edge in tens of percent's of the chord.
- Last two digits describing maximum thickness of the airfoil as percent of the chord.

This paper outlines the numeric procedure to analyze the NACA 0012 & NACA 4412 airfoil with a chord length of one meter and the Reynolds numbers of $3\text{x}10^6$. A two dimensional model is created to compare FLUENT's accuracy in the two dimensional analysis.

2. Computational Method

In this paper, the NACA 0012 and NACA 4412, the well documented airfoil from the 4-digit series of NACA airfoils, was utilized. The NACA 0012 airfoil is symmetrical; the 00 indicates that it has no camber. The 12 indicates that the airfoil has a 12% thickness to chord length ratio; it is 12% as thick as it is long. Reynolds number for the simulations was Re=3x106. The density of the air at the given temperature is ρ=1.225kg/m3 and the viscosity is μ=1.7894×10-5kg/ms. For this Reynolds number, the flow can be described as incompressible. The Reynolds average Navier-Stokes equations are solved using the green-gauss cell based gradient option and the IMPLICIT density-based solver is selected with a second order implicit transient

* Corresponding author. Tel.: +88-01713255226
E-mail address: hasanshahariar.kuet@gmail.com, rashedulhasan.047@gmail.com, mdmashud@yahoo.com

formulation for improved accuracy. The turbulent viscosity is computed through Realizable k-epsilon turbulence model with enhanced wall treatment. All solution variables were solved via second order upwind discretization scheme since most of the flow can be assumed to be not in line with the mesh.

2.1 Boundary conditions

The computational domain extended 15C upstream of the leading edge of the airfoil, 15C downstream of the trailing edge, and 20C above the pressure surface. Velocity inlet boundary condition was applied upstream (Inflow) with speed of (U=43.822 m/sec) and outflow boundary condition was applied downstream. An unstructured mesh arrangement with quadrilateral elements was adopted to map the flow domain in ground effect. [2] It involves inlet, outlet & wall boundary, the velocity components are calculated for each angle attack case as follows. The x-component of velocity is calculated by x=ucosα and the y component of velocity is calculated by y=ysinα, where α is the angle of attack in degrees. Ansys recommends turbulence intensities ranging from 1% to 5% as inlet boundary conditions. In this study it is assumed that inlet velocity is less turbulent that pressure outlet. Hence, for velocity inlet boundary condition turbulence intensity is considered 1% and for pressure outlet boundary5%. In addition, Ansys also recommends turbulent viscosity ratio of 10 for better approximation of the problem [3].

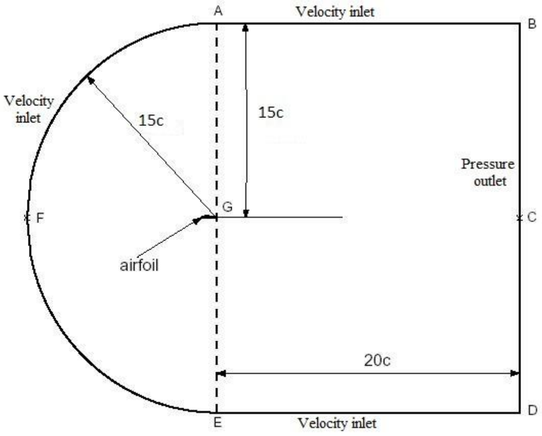

Fig 1: The dimensions and boundary conditions of the computational domain

2.2 Grid generation and Wall treatment

The grid used for simulating the NACA0012 & NACA 4412 airfoil is generated by the ANSYS Meshing is shown in Figure. The application of wall functions to modeling the near-wall region may significantly reduce both the processing and storage requirements of a numerical model, while producing an acceptable degree of accuracy.

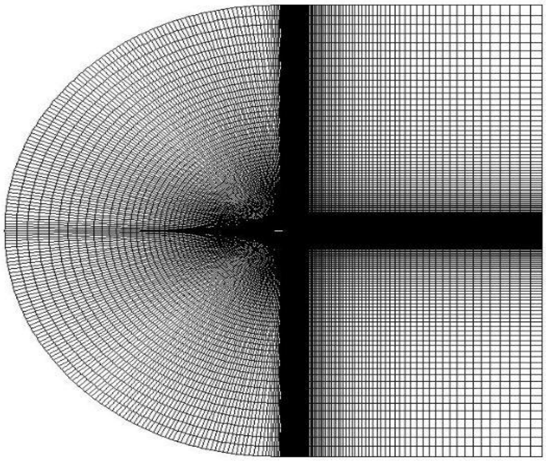

Fig 2: Mesh generation of the total structure domain

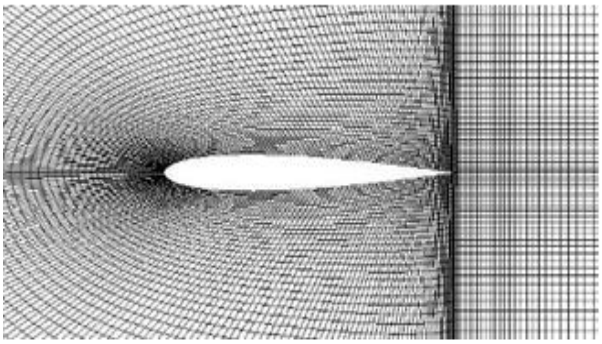

Fig 3: Mesh around NACA 0012 airfoil

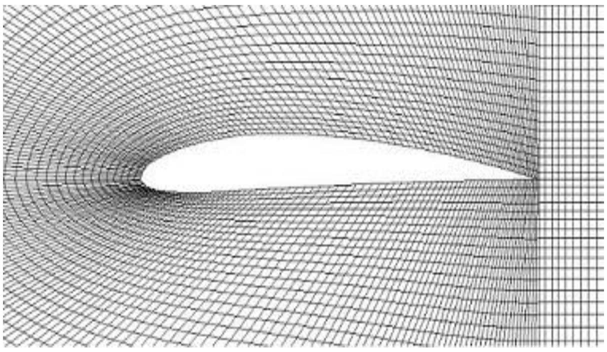

Fig 4: Mesh around NACA 0012 airfoil

To ensure sufficient boundary layer modeling, the growth rate of the inflation was set to 1.15 to give a minimum of 30 layers within the boundary layer. Beyond y+=30, the kepsilon model takes effect (due to the blending function). The non-dimensional wall parameter is defined as:

$$y^{+} = y * \frac{\sqrt{\frac{\tau_w}{\rho}}}{\mu} \quad (1)$$

* Corresponding author. Tel.: +88-01713255226
E-mail address: hasanshahariar.kuet@gmail.com, rashedulhasan.047@gmail.com, mdmashud@yahoo.com

Here, y is the distance from the wall to the centroid of the first fluid cell, μ is the local kinematic viscosity, ρ is the air density and the subscript w denotes wall properties [4]. This study revealed that a C-type grid topology with 80000 quadrilateral cells would be sufficient to establish a grid independent solution (Figure 2). The domain height is set to approximately 20 chord lengths and the height of the first cell adjacent to the surface is set to 10-5, corresponding to a maximum y+ of approximately 0.2. A y+ of this size should be sufficient to properly resolve the inner parts of the boundary layer [5].

3.3 Grid independence study

The first step in performing a CFD simulation should be to investigate the effect of the mesh size on the solution results. Generally, a numerical solution becomes more accurate as more nodes are used, but using additional nodes also increases the required computer memory and computational time. The appropriate number of nodes can be determined by increasing the number of nodes until the mesh is sufficiently fine so that further refinement does not change the results. Figure 5 shows the effect of number of grid cells in coefficient of lift at stall angle of attack (16°). This computational model is very small compared to that of NASA's validation cases.

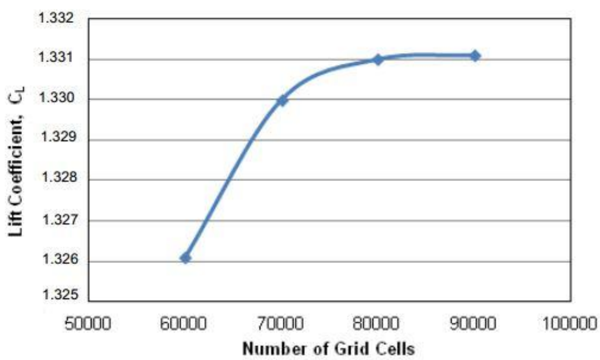

Fig 5: Variation of lift coefficient with number of grid cells [6]

3.0 Results and Discussions

After complete solution of NACA-0012 and NACA 4412 airfoil in FLUENT, resultant characteristics plot shows the effect of pressure distribution on airfoil, the training-edge pressure coefficient and pressure gradient along wall direction which suggests shock wave location and intensity. From 0 degree AoA to 16 degree AoA the lift curve is almost linear for NACA 0012. Throughout this regime no separation occurs and flow remains attached to the airfoil. At an angle of attack of roughly 15 to 16°, the flow on the upper surface of the airfoil began to separate and a condition known as stall began to develop. At stall AoA lift coefficient is reduced drastically due to intense flow separation generation. It also seen that, NACA 4412 has more lift than NACA 0012 airfoil. Also the drag co-efficient is decreases when the airfoil is camber. Figure 6 and 7 shows the variation of lift and drag co-efficient at various angle of attack. Hence it is observed that k-epsilon turbulence model with transition capabilities is predicting higher flow acceleration near the leading edge of the camber airfoil and hence relatively higher value of lift coefficient is observed. From figure 9, it is seen that the pressure distribution on upper and lower surface of NACA 4412 airfoil is more than NACA 0012 airfoil. Figures 10, 11, 12 shows the simulation outcomes of static pressure at angle of attack 8 degree with the Realizable k-epsilon turbulence model.

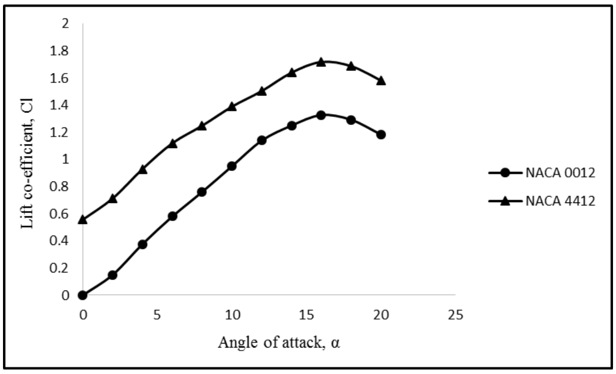

Fig 6: Lift co-efficient vs angle of attack

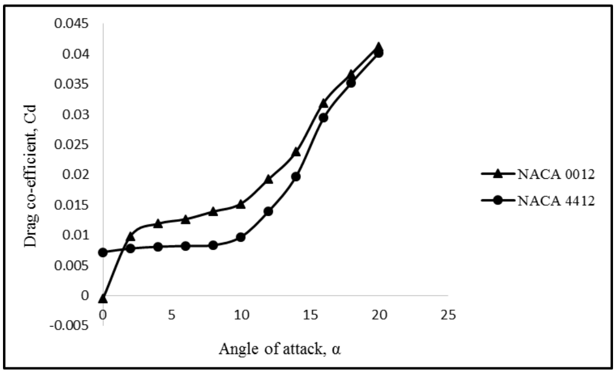

Fig 7: Drag co-efficient vs angle of attack

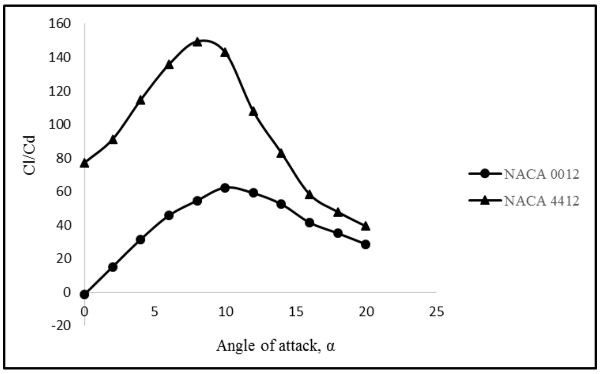

Fig 8: Lift-Drag ratio vs angle of attack

* Corresponding author. Tel.: +88-01713255226
E-mail address: hasanshahariar.kuet@gmail.com, rashedulhasan.047@gmail.com, mdmashud@yahoo.com

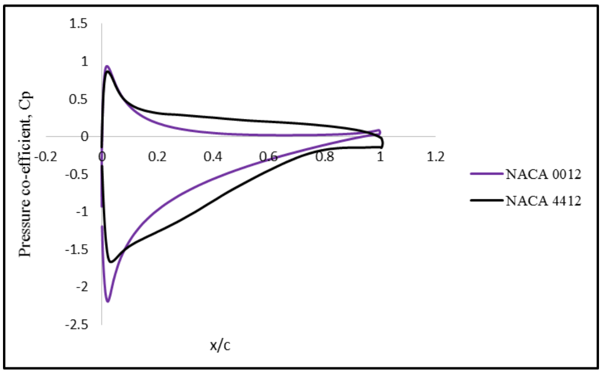

Fig 9: Pressure coefficient on the airfoil surface for 8 degree angle of attack

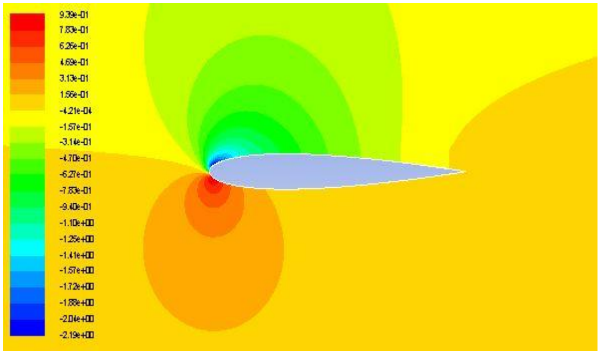

Fig 10: Contours of static pressure for 8 degree angle of attack for NACA 0012

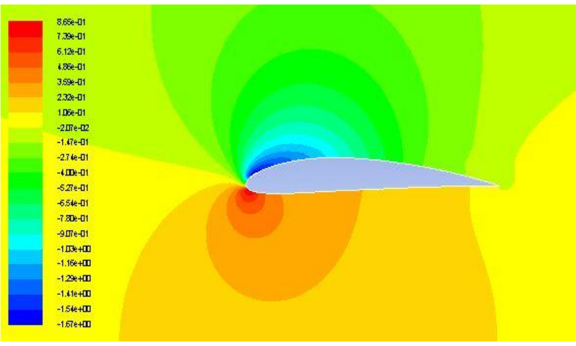

Fig 11: Contours of static pressure for 8 degree angle of attack for NACA 4412

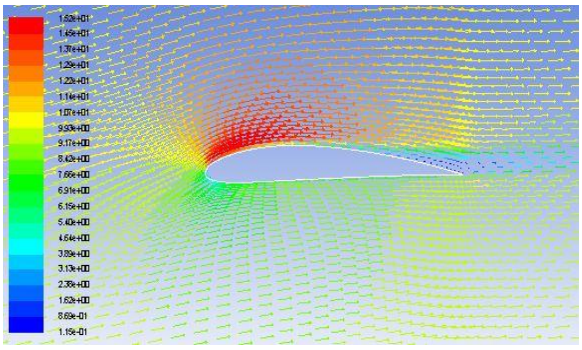

Fig 12: Velocity vectors for 8 degree angle of attack for NACA 4412

4. Conclusion

The main objective of this project is to study ANSYS FLUENT CFD software and to see the effect of aerodynamic characteristics of NACA-0012 and NACA 4412 airfoil. The flow characteristics for two-dimensional are analyzed with RANS equations and approximated by finite volume schemes with Realizable k-epsilon turbulence models. The difference in the flow characteristics of for NACA 0012 and NACA 4412 based on this study, some conclusions can be drawn as:

- Maximum lift co-efficient for NACA 0012 is 1.32803 and maximum lift co-efficient for NACA 4412 is 1.71935 at 16 degree angle of attack
- As the increase of angle of attack, drag co-efficient increases as shown in the figure
- The lift to drag ratio for NACA 0012 is 149.56 at 8 degree AOA and for NACA 4412 is 62.44 at 10 degree angle of attack

REFERENCES

[1] Ravi.H.C, Madhukeshwara.N, S.Kumarappa, “Numerical Investigation Of Flow Transition For Naca-4412 Airfoil Using Computational Fluid Dynamics”, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 7, July 2013

[2] Tousif Ahmed, Mohammad Tanjin Amin, S.M. Rafiul Islam, Shabbir Ahmed, “Computational Study of Flow Around a NACA 0012 Wing Flapped at Different Flap Angles with Varying Mach Numbers”, Global Journal of Researches in Engineering, Volume XIII Issue IV Version I, Year 2013

[3] Wilcox, David C, "Turbulence Modeling for CFD", Second edition, Anaheim, DCW Industries, 1998.

[4] C.G. Speziale and R. Abid and E.C. Anderson, "Critical Evaluation of Two-Equation Models for Near-Wall Turbulence", AIAA J., Vol. 30 No. 2, pp. 324-331(1992).

[5] P. M. Gresho and R. L. Lee and R. L. Sani, "On the Time-Dependent Solution of the Incompressible Navier-Stokes Equations in Two and Three Dimensions", In Recent Advances in Numerical Methods in Fluids, Pineridge Press, Swansea, U.K (1980).

[6] Eleni, Douvi C., Tsavalos I. Athanasios, and Margaris P. Dionissios. "Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil." Journal of Mechanical Engineering Research 4.3 (2012): 100-111

* Corresponding author. Tel.: +88-01713255226
E-mail address: hasanshahariar.kuet@gmail.com, rashedulhasan.047@gmail.com, mdmashud@yahoo.com