International Journal of Modern Trends in Engineering and Research
www.ijmter.com
e-ISSN No.:2349-9745, Date: 2-4 July, 2015

CFD Analysis of Airfoil NACA0012

Pritesh S. Gugliya[1], Yogesh R. Jaiswal[2], Akshay B. Chhajed[3], Sachin V. Jain[4], Hitesh R. Thakare[5]

[1]*Department of Mechanical Engineering, S.N.J.B's KBJ COE, Chandwad, priteshgugliya246@gmail.com,*
[2]*Department of Mechanical Engineering, S.N.J.B's KBJ COE, Chandwad, yogeshjaiswal_art@rediffmail.com,*
[3]*Department of Mechanical Engineering, S.N.J.B's KBJ COE, Chandwad, akshaychhajed126@gmail.com,*
[4]*Department of Mechanical Engineering, S.N.J.B's KBJ COE, Chandwad, jainsachin726@gmail.com,*
[5]*Department of Mechanical Engineering, S.N.J.B's KBJ COE, Chandwad, thakare.hitesh@gmail.com*

Abstract—Flow over an airfoil is important aspect of many engineering applications such as, turbines, wind mills, compressors, airplanes etc. However, experimental investigation of important flow physics characteristics around airfoil is technically and economically expensive. At this point of uncertainty, CFD can significantly help the researchers. Owing to its capability to promptly replicate the real life flow physics at comparatively lower cost, CFD can play an important role in design and optimization process. This project is aimed towards CFD analysis of subsonic flow over airfoil NACA 0012 at Reynolds number 3 x 10^6 for various values of angle of attack and Mach number. It has been observed that present CFD results are in good agreement with experimental results.

Keywords- Airfoil, angle of attack, drag force, lift force, Reynolds number

I. INTRODUCTION

Airfoil is defined as the body which, when placed in a air stream results into occurrence of an aerodynamic forces which can be utilized depending upon the application areas. These forces can be used by the wings of an aircraft, rotor, propeller blades, blades of a wind mill turbine etc. When the airfoil moves through an air stream it result into splitting up of air into two region i.e. the half of the wind passes through the upper region while the other remaining portion passes through the lower region. The upper region of an airfoil is in a curved shape while the lower portion is straight. When the air passes through the upper region it get stretched due to the shape present over there which result into the reduction of a pressure on the upper side. While the air passing from the lower region do not results into a reduction of a pressure due the present of a straight portion.

II. LITERATURE SURVEY

Murthy et al. [1] predicted the formation, growth and shedding characteristics of vortices both primary and secondary type, either flow was steady or unsteady. Lee and Jang [2] investigated experimentally, flow structure of the wake behind a NACA 0012 airfoil covered with a V-shaped micro-riblet film and the drag force acting on each airfoil were measured for Reynolds numbers ranging from Re 1.03 x10^4 to 5.14 x10^4. Shan et al. [3] presented Direct Numerical Simulation (DNS) for the flow separation and transition around a NACA 0012 airfoil with an attack angle of 4^o and Reynolds number of 10^5 based on free-stream velocity and chord length. Deng et al. [5] obtained and analyzed the details of flow separation, formation of the detached shear layer, Kelvin–Helmholtz instability and vortex shedding, interaction of nonlinear waves, breakdown, and re-attachment. Baxevanou et al. [6] reported that the most appropriate interpolation scheme for the high Re k-ε model was the Roe-Sweby upwind TVD scheme with the minmod limiter. Martinat et al. [7] provided a study of the NACA0012 dynamic stall at Reynolds numbers 10^5 and 10^6 by means of two-dimensional and three-dimensional numerical simulations as well as studied the turbulence effect on the dynamic stall by statistical modeling. Douvi et al. [8] showed the behavior of airfoil at different angles of attack and Re 3 x10^6.

Present paper is intended to report the findings of CFD study carried out on airfoil NACA 0012 for different values of angle of attack. The computational model is 2D domain using structured mesh which has been created using modeling and meshing software Gambit 2.4.6. the equations related to flow physics and turbulence have been solved using commercially available solver Fluent 6.3.26. Spalart Allmaras model has been used for the computation of turbulence in the flow field.

III. MATHEMATICAL MODELLING

3.1 Governing Equations of CFD [9]

The governing equations solved by Fluent 6.3.26 are equation of continuity, momentum, energy and equation of state, given below:-

$$\frac{\partial \rho}{\partial t} + \nabla.\ \rho\vec{v}\ = S_m \tag{1}$$

$$\frac{\partial}{\partial t}\ \rho\vec{v}\ + \nabla.\ \rho\vec{v}\vec{v}\ = -\nabla p + \nabla.\ \overline{\overline{\tau}}\ + \rho\vec{g} + \vec{F}, \tag{2}$$

$$\frac{\partial \rho h_o}{\partial t} - \frac{\partial P}{\partial t} + \mathrm{div}(\rho h_o \mathrm{U})\ =\ \mathrm{div}\ \lambda\, \mathrm{grad}\, T_s \tag{3}$$

$$P = \rho R T_s \tag{4}$$

Additionally, transport equations are also solved for different turbulence models.

3.2 Transport Equation for Spalart-Allmaras Model

$$\frac{\partial}{\partial t}\ \rho\tilde{v}\ + \frac{\partial}{\partial x_i}\ \rho\tilde{v}u_i\ = G_v + \frac{1}{\sigma\tilde{v}}\left[\frac{\partial}{\partial x_j}\left\{\ \mu + \rho\tilde{v}\ \frac{\partial\tilde{v}}{\partial x_j}\right\} + C_{b2}\rho\left(\frac{\partial\tilde{v}}{\partial x_j}\right)^2\right] - Y_v + S_{\tilde{v}} \tag{5}$$

IV. GRID INDEPENDENCE STUDY

The structure grid used in present study has been depicted in Fig. 1. Grid independence check is necessary to eliminate the erros which arise due to coarseness of the grid. In present study, grid independence of the solution has been tested using four different grid sizes i.e. with grid made up of 9000, 18032, 25296 and 33338 cells. Fig. 2 shows the effect of grid size on coefficient of lift. From Fig. 2, it is observed that value of Cl does not change significantly when grid size is increased beyond 18032 cells. Hence, further observations have been reported using grid of 18032 cells to save computational power and time.

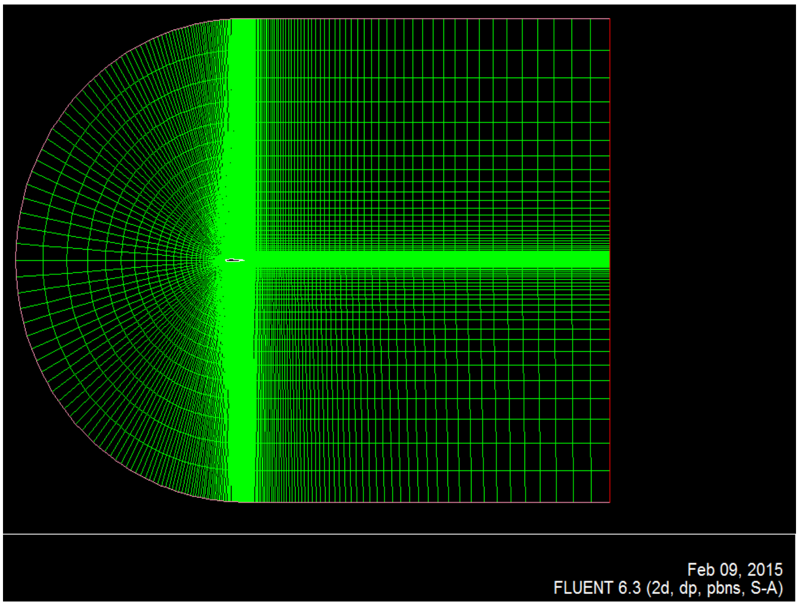

Figure 1. Structured grid used in present study

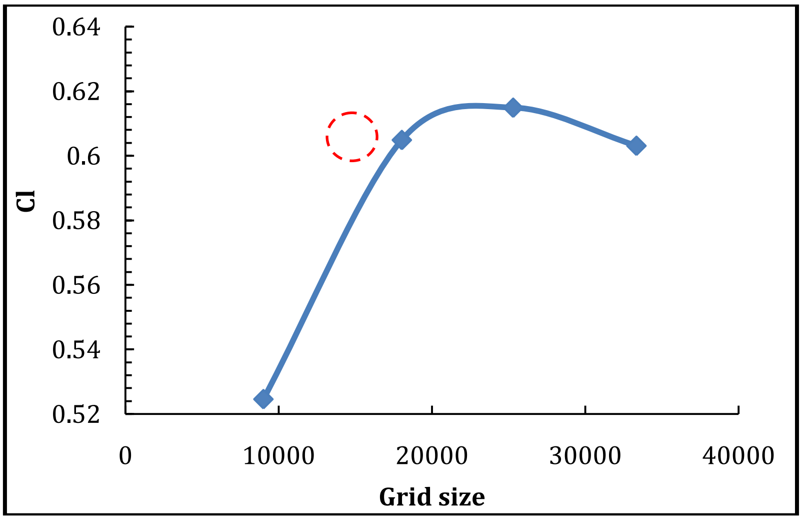

Figure 2. Effect of grid size on Cl

V. VALIDATION

Once the CFD results have been obtained, they must be compared with experimental results to check their correctness and authenticity. the results of present CFD study have been compared with experimental results of [4], as shown in Fig. 3. it is observed that for all the values of angle of attack (AOA), results of present CFD study are in good agreement with experimental results. such an accurate computational model can now be utilized to study the effect of different parameters on operating performance of airfoil.

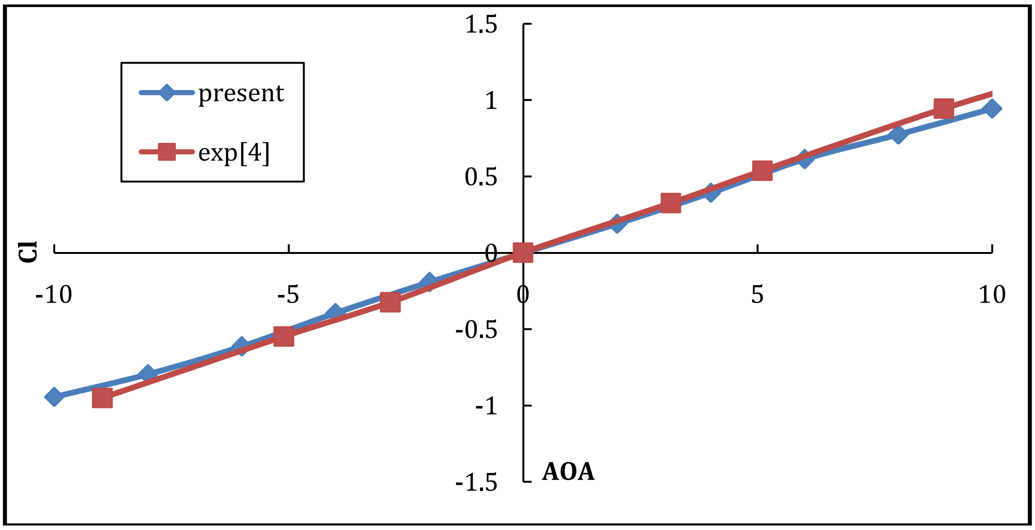

Figure 3. Comparison of experimental and CFD values of Cl

RESULTS AND DISCUSSION

Once a reliable and dependable CFD model has been established, it can be used to study the effect of various values of angle of attack on the values of drag coefficient Cd and lift coefficient Cl for Reynold Number of $3x10^6$. It has been observed during the present study that with increase in angle of attack, lift force increases substantially, however, accompanied by significant rise in drag force also. hence, attempt has been made to compare the change in drga and lift forces with change in angle of attack.

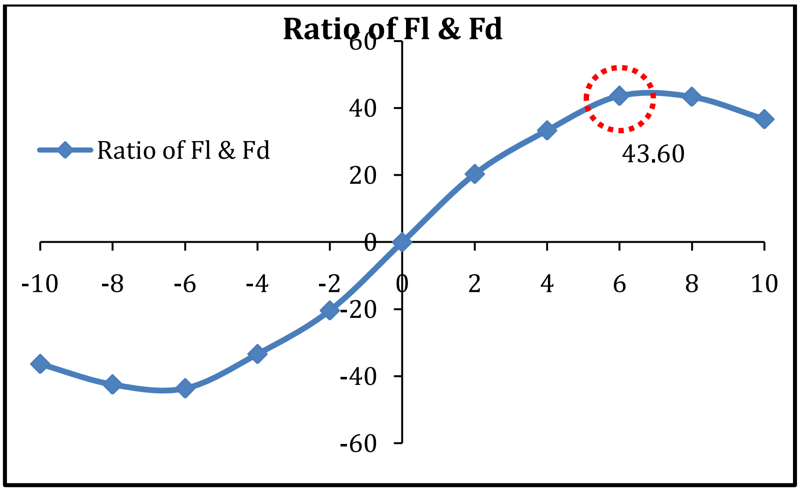

Figure 4. Variation of ratio of drag and lift force with angle of attack

Fig. 4 shows variation of ratio of lift force Fl and drag force Fd with change in angle of attack. it is oberved that this ratio increases with increase in angle of attack from 0^o to 6^o. After this, the ratio decreases with further increase in angle of attack. Maximum value of this ratio has been observed to be 43.60 at 6^o angle of attack. it means that airfoil NACA 0012 operated at angle of attack of 6^o can experience much efficient operation as compared to that at other values of angle of attack.

To gain further insights into the flow characteristics, variation of drag force experienced by airfoil with change in Mach number at two benchmark cases of angle of attack have been observed. Fig. 5 shows the variation in the magnitude of drag force with change in Mach number at angle of attack of 6^o and 15^o. Previously, in Fig. 4, it has been shown that performance is better with angle of attack of

6^o. From Fig. 5, it is observed that drag force magnitude increases abruptly when Mach number is changed for angle of attack 15^o. This causes the formation of stall at 15^o angle of attack.

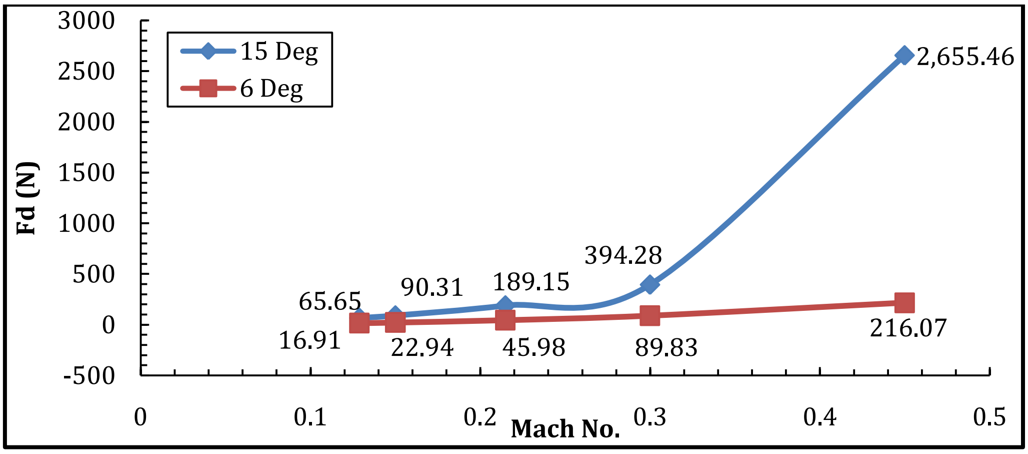

Figure 5. Variation of drag force with change in Mach number

CONCLUSION

CFD study of airfoil NACA 0012 has been carried out using Spalart Allmaras turbulence model. It is concluded that results of present CFD study are in good agreement with experimental results. furthermore, angle of attack of 6^o is best suited for airfoil operation under given condition. abrupt increase in drag force after this angle of attack results in poor performance of airfoil operation.

REFERENCES

[1] P.S. Murthy , V.S. Holla , H. Kamath Unsteady Navier-stokes solutions for a NACA 0012 airfoil , comput Method Mech Engrg. 186 (2000) 85-99.

[2] S.-J. Lee , Y.-G. Jang , Control of Flow Around a NACA 0012 Airfoil with a micro-riblet flim , department of Mechanical Engineering , Pohang University of Science and Tecnology , Pohang 790-784 , south Korea , Journal of Fluids and Structure 20 (2005) 659-672. 23May2005.

[3] Hau Shan , Li Jiang , Chaoqun Liu , Direct numerical simulation of flow sepration around a NACA 0012 airfoil , department of Mathematics , University of Texas at Arlington , P.O.Box 19408 , 411 S. Nedderman , Arlington, Tx, 76019-0408, USA, Computers and Fluids 34 (2005) 1096-1114.

[4] Douvi C. Eleni, 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, Department of Mechanical Laboratory, University of Patras, 26500 Patras, Greece, Journal of Mechanical Engineering Research Vol 4(3), pp. 100-111, March 2012 ISSN 2141-2383.

[5] Shutian Deng, Li Jiang, Chaoqun Liu, DNS for flow separation control around an airfoil by pulsed jets, Department of Mathematics, University of Texas or Arlington, Arlington, Tx 76016, USA Computers and Fluids 36 (2007) 1040-1060.

[6] C.A. Baxevanou and D.K. Fidaros Validation of numerical schemes and turbulence models combinations for transient flow around airfoil, Engineering Applications of computational Fluid Mechanics Vol. 2, No. 2, pp. 208-221(2008).

[7] G. Martinat, M. Braza, Y. Hoarau, G. Harran, Turbulence modeling of the flow past a pitching NACA 0012 airfoil at 105 and 106 Reynolds Numbers, Journal of fluids and structures 24 (2008) 1294-1303.

[8] Eleni C. Douvi, Athanasios I. Tsavalos, Dionissios P. Margaris, CFD calculation of the flow over a NACA 0012 airfoil, Fluid Mechanics Laboratory University of Patras, Department of Mechanical Engineering and Aeronautics 26500 Patras, Greece, July2010.

[9] FLUENT 6.3, Theory, Fluent Inc., September 2006.

