Volume-4, Issue-4, April-2015 • ISSN No 2277 - 8160

GLOBAL JOURNAL FOR RESEARCH ANALYSIS International

Research Paper | Engineering

An Analysis of Pressure Difference on Airfoil by Changing Angle of Attack and Gurney Flap Height

Hardik S. Patel Dept. of mechanical engineering LDRP-ITR, KSV Gandhinagar, India

ABSTRACT

The present study comprises steady state, two-dimensional computational investigations performed on NACA0012 (National advisory committee for aeronautics) airfoil to analyse the effect of pressure difference on airfoil with GF (gurney flap) at various angle of attack using viscous-laminar model of FLUENT & using CAD preparation of two dimensional NACA 0012 airfoil. Airfoil with GF is analysed for four different heights from 0%to 6% of the chord length and three angles of attack from 0◦ to 16◦. Static pressure distribution on the airfoil surface is present. From computational investigation, it is recommended that Gurney flaps with a approximately height of 4% chord to be installed perpendicular to chord and on trailing edge as possible to obtain maximum pressure.

KEYWORDS : NACA 0012 airfoil; Gurney flap; CFD analysis; Attack angle; Pressure difference

I. Introduction

The aerodynamic characteristics of airfoils at a small chord Reynolds number (less than 5x105) are becoming increasingly important from both fundamental and industrial point of view, due to recent developments in small wind turbines, small unmanned aerial vehicles, micro-air vehicles, as well as researches on bird/insect flying aerodynamics. It is usually not enough to optimize one single airfoil shape. The need arises for conformal changes of the airfoil to achieve best performance in both conditions. This is where high lift devices enter the design space. Therefore, study and analysis of these devices play a dominant role in design. The high-lift systems have been studied for many years since these systems play a major role in economic success of an aircraft. An effective high-lift system allows lower take-off and landing speed, greater payload capacity of given wing and longer range for a given gross weight. So, high-lift aerodynamics continues playing an important role in the design of a new aircraft. Hence, there is a continuous need for improving the maximum lift.

CFD study of airfoils to predict its visualisation and surveillance of flow field pattern around the body. Wing with flap is usually known as high lift device. CFD facilitates to envisage the behavior of geometry subjected to any sort of fluid flow field. This fast progression of computational fluid dynamics (CFD) has been driven by the necessity for more rapid and more exact methods for the calculations of flow fields around very complicated structural configurations of practical attention. CFD has been demonstrated as an economically viable method of preference in the field of numerous aerospace, automotive and industrial components and processes in which a major role is played by fluid or gas flows. In the fluid dynamics, for modelling flow in or around objects, many commercial and open source CFD packages are available. The computer simulations can model features and details that are tough, expensive or impossible to measure or visualize experimentally. it is very important to understand the characteristics of the wing having different flap angles. This study does not provide any experimental data for the flow over the flapped airfoil. Selecting a proper turbulence model, the structure and use of a model to forecast the effects of turbulence, is a crucial undertaking to study any sorts of fluid flow. It should model the whole flow condition very accurately to get satisfactory results. Selection of wrong turbulence model often results worthless outcomes, as wrong model may not represent the actual physics of the flow. Turbulent flow dictates most flows of pragmatic engineering interest.

Main goal of this study is to do parametric analysis of flow over (National Advisory Committee for Aeronautics) NACA 0012 airfoil with plain flap at various angles of attack and flap heights. The measurements were carried out for the Reynolds number of 3x10^5 and attack angle of 0 to 16 with 2 intervals and flap height of 0 to 6%c with 4 intervals to investigate the effects of these parameters on aerodynamic characteristics of NACA0012 airfoil. Hence the present investigation is undertaken, computationally to see the influence of these parameters on the airfoil performance by using CFD software.

II. Theoretical background

It was first used by Dan Gurney on the top trailing edge of the rear wing on his race car to provide extra rear end down force with minimal aerodynamics disturbance [1]. Liebeck [2] conducted first wind tunnel experiments on GF. An excellent review of GF research for aircraft wings and other aerodynamics applications was presented by Wang et al. [3]. Jang et al. [4], Yoo [5], and Li et al. [6] have verified the lift enhancement of GF in their experiments. Neuhart and Pendergraft [7] visualized recirculation zones behind GF in their water tunnel experiments and also recommended to keep the GF height less than 2% of the chord length to reduce drag penalty which was also verified by Myose et al. [8]. Experiments on GF for GU25-5(11)-8 airfoil by Galbraith [9] concluded that GF should be mounted at distance $S <$ 10% to prevent major performance degradation as verified by Li et al. [10] for NACA 0012 airfoil. Brown and Filippone [11] conducted experiments at Reynolds number ranging from Re = 42000 to 1.6 × 105. Their analysis also shows that the optimum height of GFs is always below the boundary layer thickness at the trailing edge. Lianbing et al.[12] have investigated performance of wind turbine NACA0012 airfoil using FLUENT programs. Spalart Allmaras turbulence model to numerical solutions was used by Lianbing et al. of airfoil at 3×106 Reynolds number for lift and drag performance and stall angle. Troolin et al.[13], added Gurney flap with NACA 0015 airfoil and they numerically investigated performance of this new design. They saw that lift coefficient increased but drag coefficient was not change so this design was useful.

III. Computational method

After The well documented airfoil, NACA 0012, is utilized in this study. The free stream temperature is 288.16 K, which is the same as the ambient temperature. The density of the air at the given temperature is ρ = 1.225 kg/m^3 and the dynamic viscosity is 1.7894×10-5 kg/ms. Reynolds number for the simulations is Re=3x10^5, flow for this Reynolds number can be labeled as incompressible[14]. This is a supposition close to reality and there is no necessity to resolve the energy equation[15]. The flow is considered to be viscid, incompressible and steady, and the uniform flow velocity is taken as 43.82 m/s. A segregated, implicit solver, ANSYS Fluent 14.5, is utilized to simulate the problem. The airfoil profile is engendered in the Design Modeler (see Fig. 1) and using CAD software Creo (see Fig.2), importing co-ordinates from airfoil tools[16] (see Fig.3) , boundary conditions, meshes are created in the pre-processor FLUENT[17] . The resolution and density of the mesh is greater in regions where superior computational accuracy is needed, such as the near wall region of the airfoil (see Fig.5).

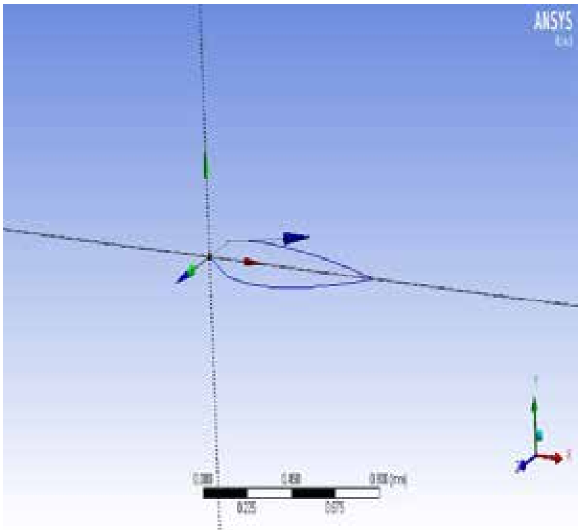

Fig. 1. Airfoil profile generated in design modeller

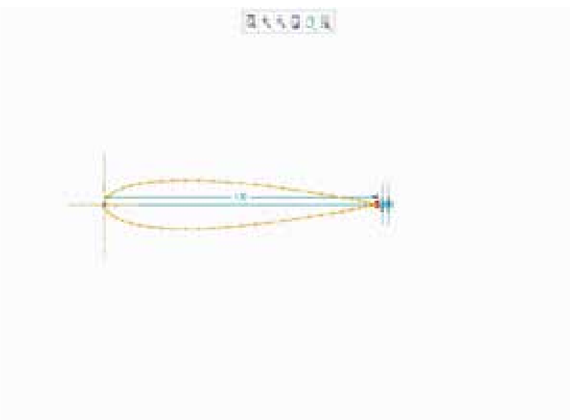

Fig. 2. 2-D airfoil sketch in Creo

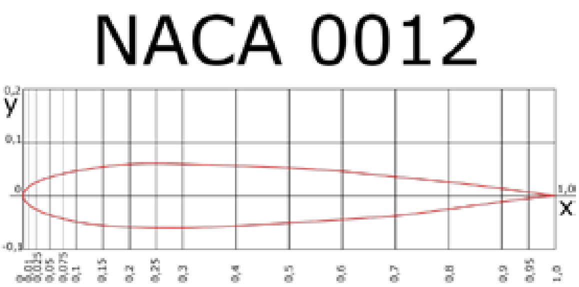

Fig. 3. Plotted Co-ordinates of NACA 0012[16]

As the first step of accomplishing a CFD simulation the influence of the mesh size on the solution results should be investigated. Mostly, more accurate numerical solution is obtained as more nodes are used, then again using added nodes also escalates the requisite computer memory and computational time. The determination of the proper number of nodes can be done by increasing the number of nodes until the mesh is satisfactorily fine so that further refinement does not change the results[14]. C-type grid topology is applied to establish a grid independent solution. As shown in Fig.6 number of nodes are 43000 and type of cells are quadrilateral. This domain represents a free stream region around a NACA 0012 airfoil (see Fig.4). In the working domain, four boundaries have been specified. Inlet is considered as velocity inlet and Outlet is considered as pressure outlet while wall and Airfoil are considered as no-slip wall.

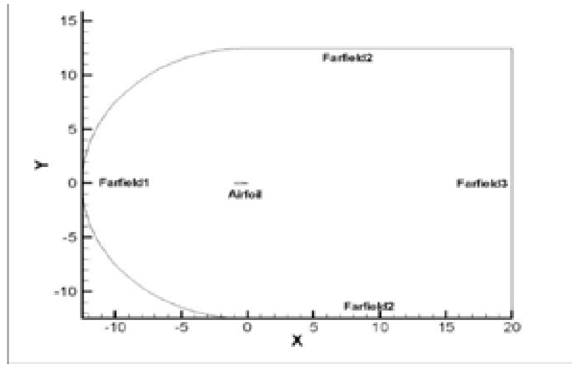

Fig. 4. Boundary domain

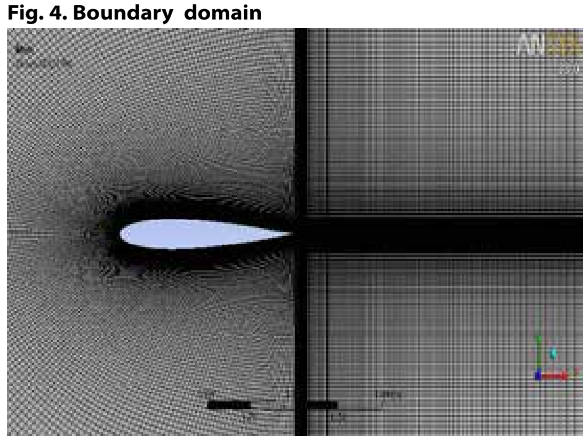

Fig. 5. Highly dense mesh near airfoil

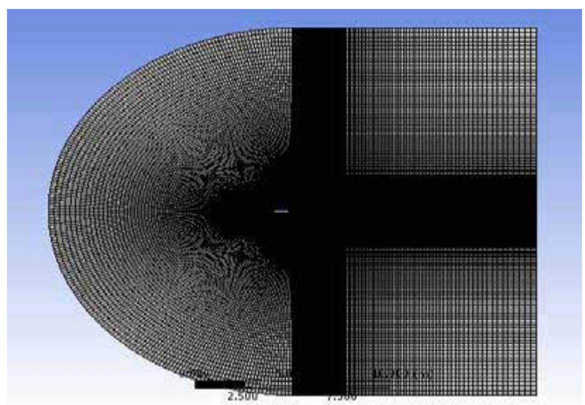

Fig. 6. C-type grid domain

IV. Result and discussion

Pressure based solver utilizing viscous-laminar model in Fluent facilitates mimicking compressible flow over the body. Flow having Mach number less than 0.3 is considered incompressible. Angle of attack values are tried between 0 and 16 degrees in steps of 8 degree for 0 to 6% gurney flap in step of 2 degrees. The static pressure for the gurney flap cases for AOA=0 degree is given in Figure 7 to 10 for comparison with the clean airfoil.

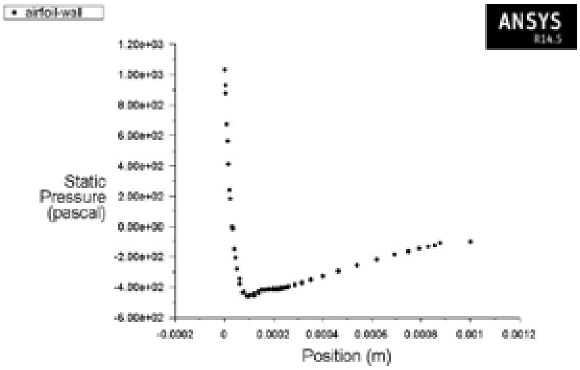

Fig. 7. Static pressure vs. position in case of 0 AOA & 0%c of flap height

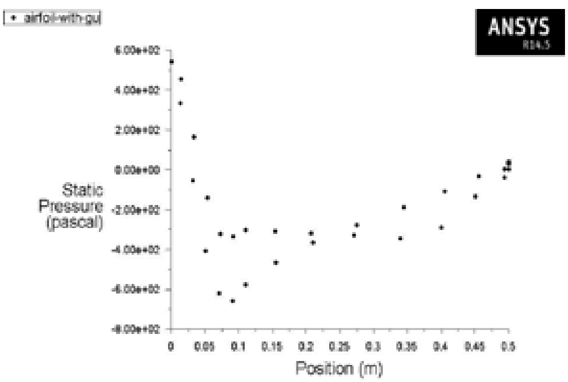

Fig. 8. Static pressure vs. position in case of 0 AOA & 2%c of flap height

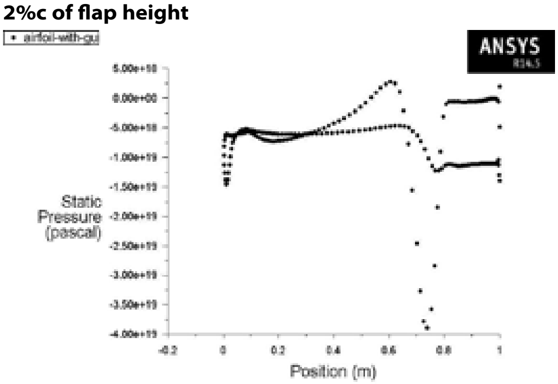

Fig. 9. Static pressure vs. position in case of 0 AOA & 4%c of flap height

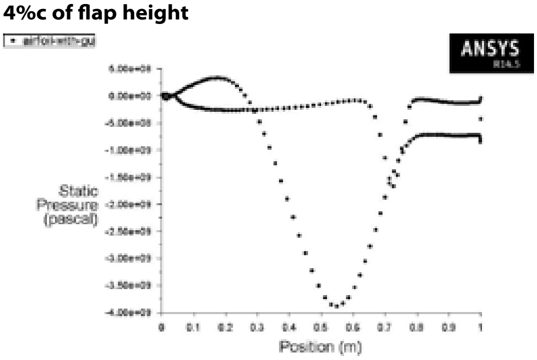

Fig. 10. Static pressure vs. position in case of 0 AOA & 6%c of flap height

Here positive (+) sign in static pressure shows negative pressure on suction (upper) side & negative (-) sign shows positive pressure on pressure (lower) side of an airfoil. As it can be seen from the figures, pressure distribution is changing seriously near the gurney flap region around the trailing edge which creates lift force even in zero angle of attack. After the gurney flap, it is expected that vorticities should occur according to the literature [2]. The vorticities are clearly seen behind the flap in the trailing edge in figures. The pressure difference can be seen in figures. Especially for the case of gurney flap with 2%c length, pressure difference is much more than the other cases. And the vorticities are more distinct which causes more positive pressure. However the positive pressure increases with the angle of attack. Simulations for various angles of attack are done in order to be able to observe the results for the different flap heights. As shown in figures 7 & 11, it could be observed that at low angles of attack, the pressure difference increases linearly with angle of attack. Flow is attached to the airfoil throughout this regime. Behavior of static pressure of an airfoil with 16 AOA & 0%c flap height is seen approximately similar to that of figure 11. At AOA=0, the pressure variation over the suction and pressure side of the airfoil showed a symmetric distribution, as expected, shown in figure 7. At an angle of attack of roughly around 16 degree, the flow on the upper surface of the airfoil begins to separate and a condition known as stall begins to develop. The actual airfoil has laminar flow over the forward half. In order to get more accurate results, the computational domain could be splitted into two different domains to run mixed laminar and turbulent flow. The disadvantages of this approach are that the accuracy of simulations depends on the ability to accurately guess the transition location, and a new grid must be generated if the transition point has to change[19] [20]. The pressure on the lower surface of the airfoil is greater than that of the incoming flow stream and as a result it effectively "pushes" the airfoil upward, normal to the incoming flow stream. On the other hand, the components of the pressure distribution parallel to the incoming flow stream tend to slow the velocity of the incoming flow relative to the airfoil, as do the viscous stresses. The trailing edge stagnation point moves slightly forward on the airfoil at low angles of attack and it jumps rapidly to leading edge at stall angle. A stagnation point is a point in a flow field where the local velocity of the fluid is zero. The upper surface of the airfoil experiences a higher velocity compared to the lower surface. That was expected from the pressure distribution. As the angle of attack increases the upper surface velocity is much higher than the velocity of the lower surface.

Pressure values increased with attack angle as smaller Reynolds number results. The separation and reattachments couldn't determine accurately from the pressure distribution due to the fact that the pressure tappings cannot be completely flush along the pressure and suction side of the airfoil. As the flap height increases pressure distribution over the airfoil increases thus the lift force generated by the airfoil also increases. Therefore, by increasing height of the flap, even without changing the AOA of the airfoil, we can increase the lift over an airfoil.

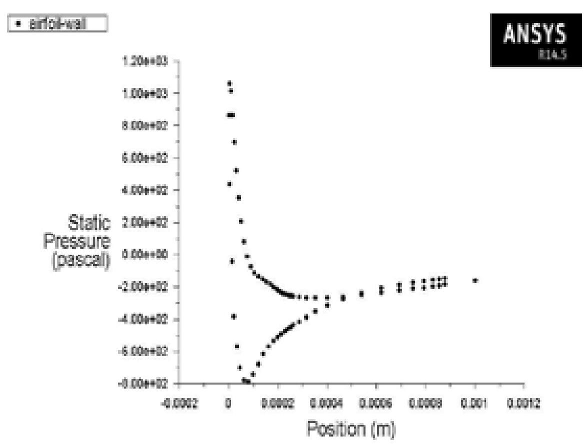

Fig. 11. Static pressure vs. position in case of 8 AOA & 0%c of flap height

V. CONCLUSION

Present study divulges behavior of NACA 0012 airfoil at different flap heights and angle of attacks. Pressure distribution curves show that Gurney flap increases upper surface suction and lower surface high pressure which results in lift enhancement and hence their aerodynamic performance can be significantly improved. Gurney flap turn the flow on a blade towards the direction of the suction surface. The pressure coefficient of the suction side of the airfoil initially increased near the leading edge and then showed a monotonously decrease up to trailing edge for all angle of attack. A symmetric pressure distribution was obtained along the suction and pressure side of the airfoil at zero attack angles. Maximum pressure is obtained at 4%c even when angle of attack is zero. Pressure difference is higher when angle of attack is 8. The Gurney flap effect is interpreted as a special camber effect or effective camber effect. Turbulence modeling is also very important for the accuracy of the results. This comprehensive study will facilitate efficient design of wing sections of aircrafts and an optimized flight.

Acknowledgment

Hardik S Patel is supported by department of Mechanical Engineering. The author would like to thank K. H. Bhavsar and A. R. Patel of LDRP-ITR for their support and discussions. Technical assistance was also given by them. Assistance appreciated, in addition to all staff at the University of Kadi Sarva Viswavidyalaya

REFERENCES

[1] C. S. Jang, J. C. Ross, and R. M. Cummings, "Numerical investigation of an airfoil with a Gurney flap," Aircraft Design, vol. 1, no. 2, pp. 75–88, 1998. | [2] R. H. Liebeck, "Design of subsonic aerofoils for high lift," Journal of Aircraft, vol. 15, no. 9, pp. 547–561, 1978. | [3] J. J. Wang, Y. C. Li, and K.-S. Choi, "Gurney flap-Lift enhancement, mechanisms and applications," Progress in Aerospace Sciences, vol. 44, no. 1, pp. 22–47, 2008. | [4] C. S. Jang, J. C. Ross, and R. M. Cummings, "Computational evaluation of an airfoil with a Gurney Flap," AIAA Paper 92–2708, 1992. | [5] N.-S. Yoo, "Effect of gurney flap on NACA 23012 airfoil," KSME International Journal, vol. 14, no. 9, pp. 1013–1019, 2000. | [6] Y. Li, J. Wang, and P. Zhang, "Effects of Gurney flaps on a NACA0012 airfoil," Flow, Turbulence and Combustion, vol. 68, no. 1, pp. 27–39, 2002. | [7] D. H. Neuhart and O. C. Pendergraft Jr., "A water tunnel study of Gurney flap,"Tech. Rep. TM-4071, NASA, 1988. | [8] R. Myose, I. Heron, and M. Papadakis, "Effect of Gurney flaps on a NACA 0011 Airfoil," AIAA Paper 96-0059, 1996. | [9] R. A. M. Galbraith, "The aerodynamic characteristics of a GU25-5(11)8 aerofoil for low Reynolds numbers," Experiments in Fluids, vol. 3, no. 5, pp. 253–256, 1985. | [10] Y.Li, J.Wang, andP. Zhang, "Influences ofmounting angles and locations on the effects of Gurney flaps," Journal of Aircraft, vol. 40, no. 3, pp. 494–498, 2003. | [11] L. Brown andA. Filippone, "Aerofoil at low speeds with Gurney flaps," Aeronautical Journal, vol. 107, no. 1075, pp. 539–546, 2003. | [12] L. B. Li, Y. W. Ma, and L. Liu, "Numerical simulation on aerodynamics performance of wind turbine airfoil," in Proc. World Automation Congress (WAC), 2012, pp. 1-4. | [13] D. R. Troolin, E. K. Longmire, and W. T. Lai, "Timere solved PIV analysis of flow over a NACA 0015 airfoil with Gurney flap," Experiments in Fluid, vol. 41, pp. 241-254, April 2006. | [14] S. Salcedo, F. Monge, F. Palacios, F. Gandia, A. Rodriguez, and M. Barcala, "Gurney flap and trailing edge devices for wind turbines," in Scientific Proceedings of the EuropeanWind Energy Conference & Exhibition (EWEC '06), Athens, Greece, 2006 | [15] Tousif Ahmed, Md. Tanjin Amin, S.M. Rafiul Islam, "Computational Study of Flow Around a NACA 0012 Wing Flapped at Different Flap Angles with Varying Mach Numbers" in Global Journal of Researches in Engineering, Volume 13 Issue 4 Version 1.0 Year 2013 | [16] http://www.mh-aerotools.de/airfoils/jf_applet.htm | [17] http://www.fluent.com/software/sf_mesh_and_tutorials/tutorial_airfoil.htm | [18] http://www.sti.nasa.gov/disclaimers/ | [19] Silisteanu P.D., Botez R.M, (2010), "Transition flow occurrence estimation new method", 48th AIAA Aerospace Science Meeting including The New Horizons Forum and Aerospace Exposition, Orlando, Florida, Etats - Unis, 7-10 janvier | [20] Launder, B.E., Spalding, D.B. (1974), "The numerical computation of turbulent flows", Computer Methods in Applied Mechanics and Engineering, Vol. 3, pp.269-89. | | |