6th International Conference on Recent Trends in Engineering, Science & Management
Shri Guru Teg Bahadur Khalsa College Anandpur Sahib, Punjab (ICRTESM-17)
8th January 2017 , www.conferenceworld.in ISBN: 978-93-86171-21-4

EFFECT OF REFLECTED TRAILING EDGE ARC ON LIFT AND DRAG COEFFICIENT ON NACA4412 AIRFOIL

Sameer Pithiya[1], Ramesh Bhoraniya[2]

[1,2] *Mechanical Engineering Department, Marwadi education foundation group of institution, (India).*

ABSTRACT

Effect of reflected trailing edge arc of NACA4412 airfoil is carried out by analysis of airfoil at zerodegree angle of attack. As the length of the arc is increase there is a significant drop of lift coefficient. We started with Geometrical meshing in Gambit and Grid independent solution for geometry. The main objective of this study is to explore the effect of the trailing edge arc on coefficient of lift and drag with the help of ANSYS (FLUENT). This phenomena is used for decreasing lift force at given angle of attack and used during a landing of aircraft and in the racing cars for increase the downforce so that increase the drive speed of car.

Keywords: ANSYS (Fluent), Drag coefficient, GAMBIT, Lift coefficient, NACA4412.

I. INTRODUCTION

Could it be imagined how an aircraft gets in to air? It goes this way i.e. when the earliest serious work on the development of the airfoil sections began in the late 1800's. Although it was just known that flat plates would produce lift when set at an angle of incidence some suspected that the shapes with curvature that more closely resembled the bird wings would produce more lift or do so, more efficiently. Sir H.F.Phiips patented a series of airfoil designs when he tested them in his earliest wind tunnel experiments.

An airfoil (in American English) or an aerofoil (in British English) is the shape of a wing or blade in cross section. An Airfoil shaped body moved through a fluid produces an aerodynamic force. The component of this force perpendicular to the direction of motion is called lift. The component parallel to direction of motion is called drag. The lift of an airfoil is primarily the result of angle of attack and shape (in particular its chamber). Airfoil design is a major facet of aerodynamics. Airfoils are more efficient lifting shapes able to generate more lift (up to a point) with less drag. This has to be the most primary and basic characteristics to be kept in mind while designing an airfoil for a wing span. Various airfoils serve different flight regimes. Asymmetric airfoils can generate lift at zero angle of attack while a symmetric airfoil may better suit frequent inverted flight as in aerobatic airplane.[1]

1.1 Airfoil nomenclature

Airfoil geometry can be characterized by the coordinates of the upper and lower surface. It is often summarized by a few parameters such as: maximum thickness, maximum camber, position of max thickness, position of max camber, and nose radius. One can generate a reasonable airfoil section given these parameters. This was done by .Eastman Jacobs in the early 1930's to create a family of airfoils known as the NACA Sections.

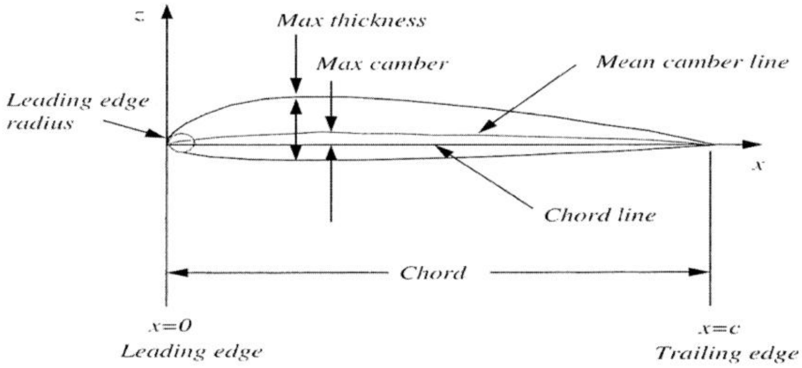

Figure 1 nomenclature of airfoil

The lift force and drag force are defined by this equations.[5]

$$\text{Lift force,} \quad L = \frac{1}{2}\rho A v^2 C_l \qquad \ldots\ldots (1)$$

$$\text{Drag force,} \quad L = \frac{1}{2}\rho A v^2 C_d \qquad \ldots\ldots (2)$$

Here , C_l and C_d are lift and drag coefficient, ρ is density of air, v is velocity of air and A is area of wing span which is $1m^2$ for the two dimensional geometry.

In 2008, Justin L. Smith, Henry Z. Graham, and James E. Smith studied the aerodynamics of airfoil in ground effect by using a two-dimension CFD analysis. They studied a effect on lift and drag on airfoil at different height from the ground. They calculate at different height to chord ratio 0.05, 0.3, 0.4, 0.6, 0.8 and 1.

The main objective of this study is to explore the effect of the trailing edge arc on coefficient of lift and drag with the help of ANSYS (FLUENT). This phenomena is used for decreasing lift force at given angle of attack and used during a landing of aircraft and in the racing cars for increase the down force so that increase the drive speed of car.

II. METHODOLOGY

The geometry of the airfoil was obtained from airfoil data site and the grid was generated by GAMBIT. The computational simulation done by FLUENT solver using Finite Volume Approach. After that aerodynamic characteristic of reflected trailing edge arc.

The NACA 4412 airfoil 2-D profile was obtained by importing the airfoil coordinates to GAMBIT. The 2-D unstructured quadrilateral mesh was utilized for computing the flow around the model. The airfoil geometry placed in a rectangular domain in which the boundary condition for left side is inlet velocity, right side is pressure outlet and top and bottom wall is given as a free stream condition shown in fig. 6.

The number of nodes created in airfoil geometry on upper and lower surface are 400 and in rectangular domain, number of node at inlet and outlet is 300 and top and bottom is 400 after grid independent study. Fig. 7 shows a close up of grid structure of naca4412 airfoil at trailing edge deflection of length 12% of chord length.

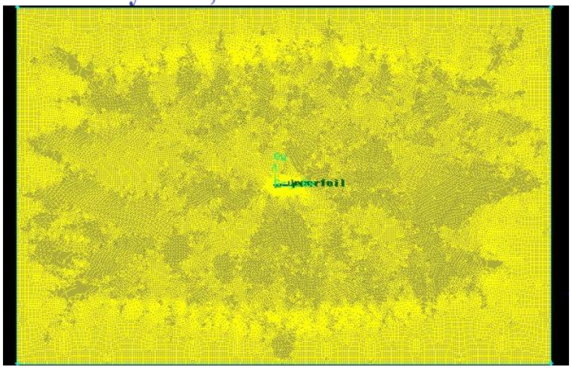

Figure 2 airfoil with rectangular domain

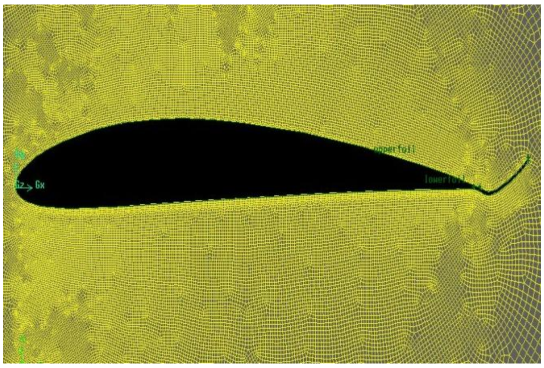

Figure 3 airfoil with reflected trailing edge arc 12%C

The air velocity is 30 m/sec and K-ω SST turbulence model used for the analysis. Gradient method used is green gauss node base because it is more accurate for aerodynamic flows. Boundary condition in fluent for airfoil, at the top and bottom wall the free stream condition, at inlet the velocity inlet, at outlet the pressure outlet and upper and lower surface of the airfoil is wall at no slip condition.

III. VALIDATION WORK

The research work that was chosen to validation case is the validation of an airfoil in ground effect regime using 2-D CFD analysis by Justin Smith, Henry Graham, and James Smith. [2] In this paper researchers do a analysis work on ansys for the analyze the aerodynamics characteristics of the NACA4412 airfoil while in ground effect flight. This research work is chosen because of similar NACA airfoil was chosen in current research.

The geometry creation and mesh generation process are done in software GAMBIT. This software provide direct interface to the flow modeling software ANSYS FLUENT. The grid structure was created using an unstructured mesh. The grid size or numbers of grid are also determined after the grid independent study for accurate results. The grid structure was created to be tested in FLUENT at 7 different height to chord ratios: 0.05, 0.15, 0.3, 0.4, 0.6, 0.8 and 1 at angle of attack 0 degree.

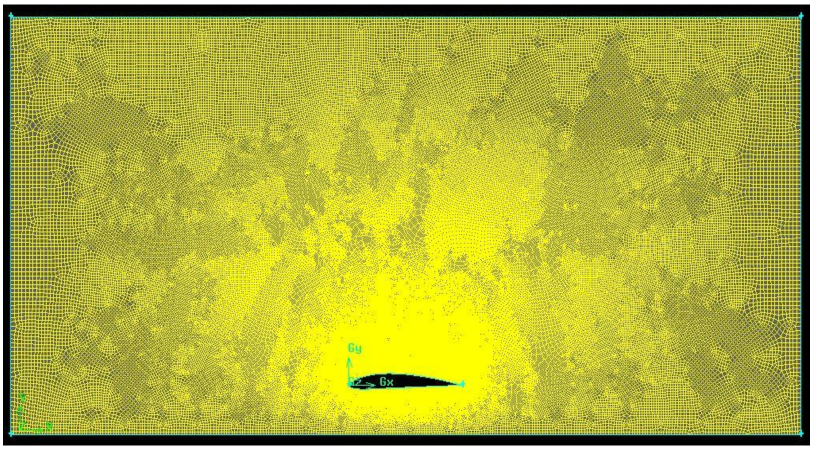

Figure 4 Close up of grid around NACA4412 at 0^0 AOA at H/C 0.4

The grid was created by unstructured mesh and size of the grid is determined after grid independence test.[3] The number of grid points on each upper and lower surface is 400 which is consistent for each setup. Solution criteria are used is same as the researcher. The lift coefficient and drag coefficient at different height are as shown below.

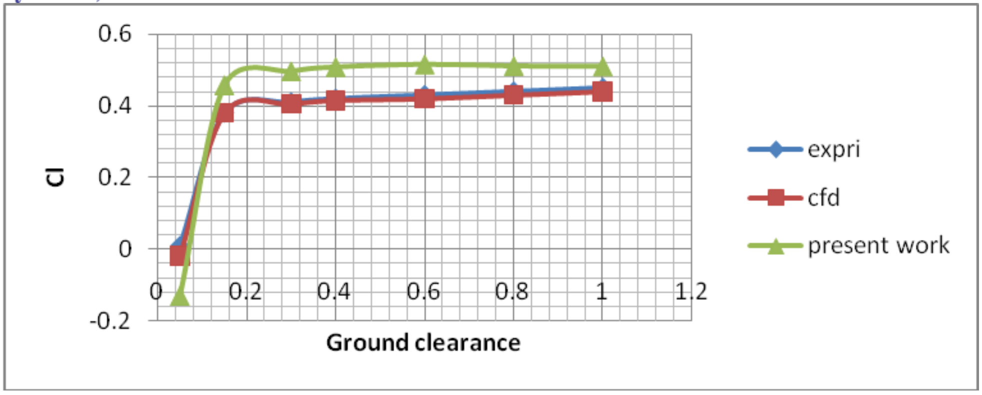

Figure 5 Coefficient of Lift at different height at 0^0 AOA

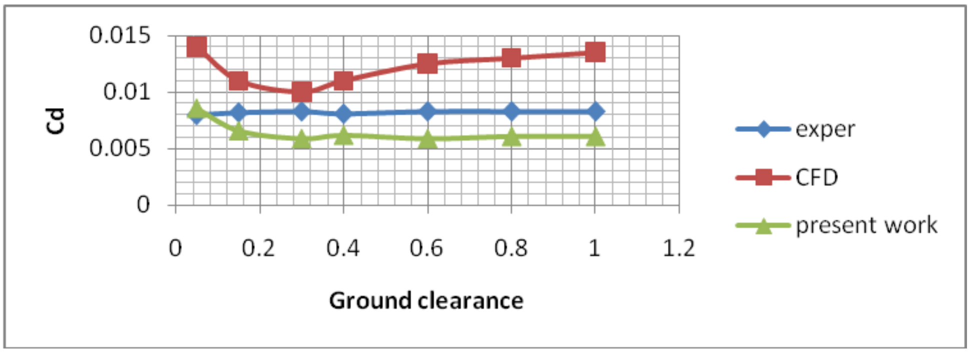

Figure 6 Coefficient of Drag at different height at 0^0 AOA

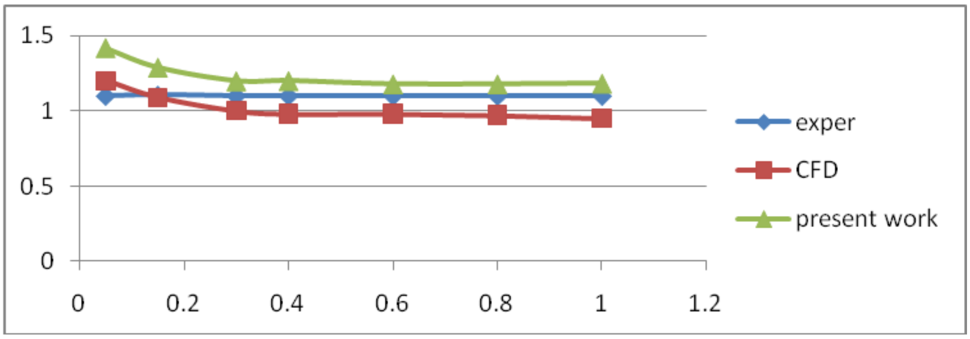

Figure 7 Coefficient of Lift at different height at 6^0 AOA

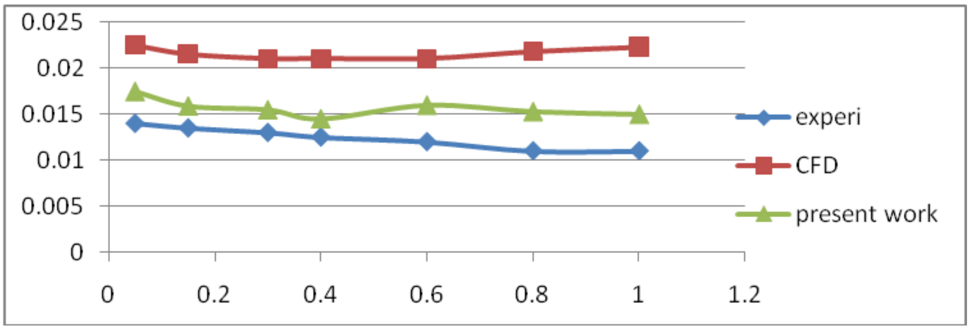

Figure 8 Coefficient of Drag at different height at 6^0 AOA

IV. RESULTS AND DISCUSSIONS

The lift and drag coefficient of the different reflected arc length is observed and also pressure coefficient at upper and lower surface are observed at 0^0 Angle of attack is observed. The trailing edge reflector arc length is consider in this case is length of 4%, 6%, 8%, 10% and 12% of Chord length. In our case chord length is taken as 1m.

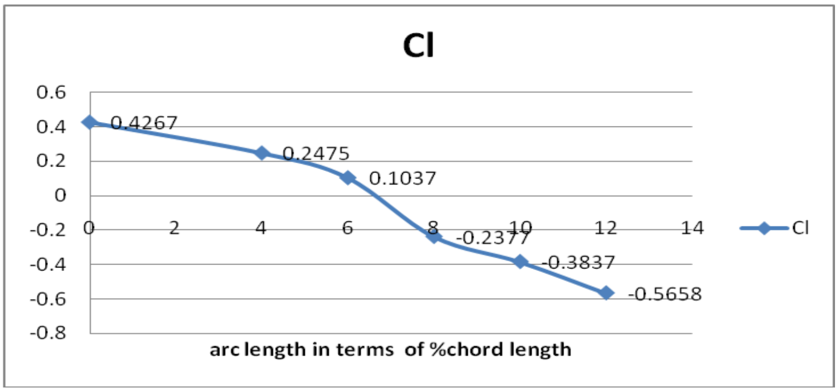

Figure 9. Arc length as % of chord length vs lift coefficient

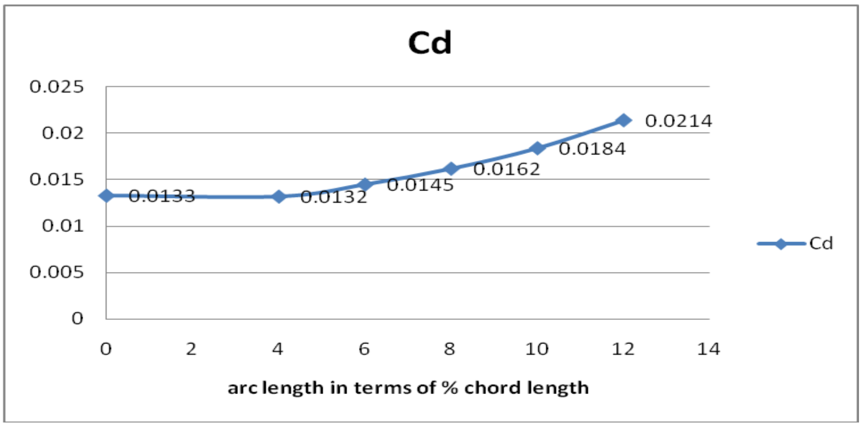

Figure 10. Arc length as % of chord length vs drag coefficient

The lift force is due to the effect of pressure difference between upper and lower airfoil. When the air passes through airfoil the pressure at the lower side of the airfoil is higher and upper side of the airfoil is low, due to this the airfoil exert force at the direction of higher to lower pressure so that lift generated. When the difference between the both upper and lower side of airfoil is more than lift force is more[4]. In fig.11 the simple airfoil pressure coefficient is obtained by fluent, ii conclude that the more lift force is obtained by this simple geometry as compared to other geometry with reflector arc at trailing edge with different length as shown in fig.12 to fig.16.[6]

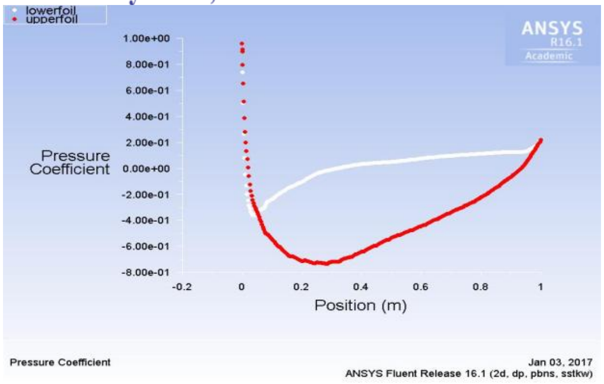

Figure 11. Cp of naca4412

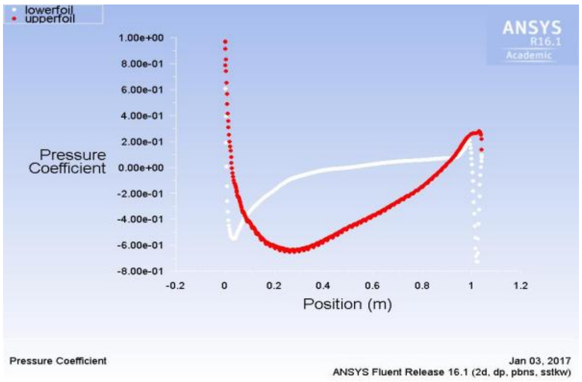

Figure 12. Cp with reflected arc length 4%C

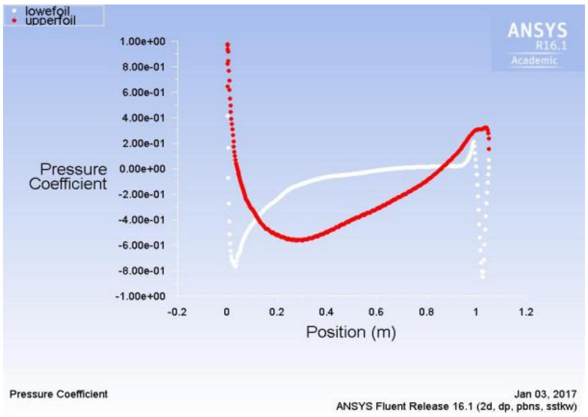

Figure 13. Cp with reflected arc length 6%C

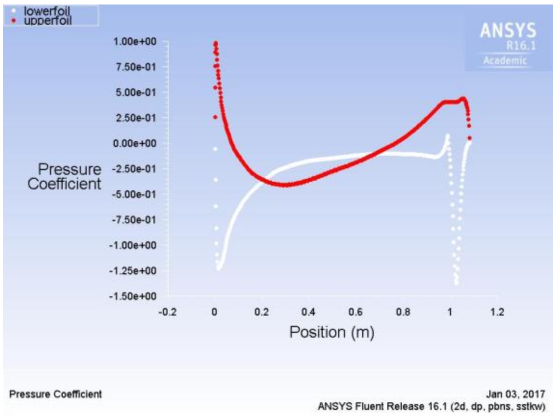

Figure 14. Cp with reflected arc length 8%C

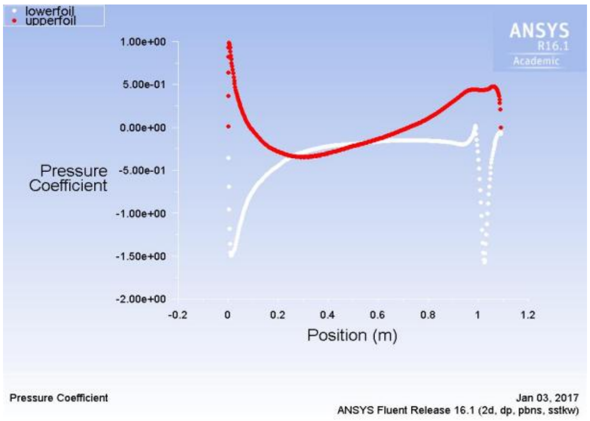

Figure 15. Cp with reflected arc length 10%C

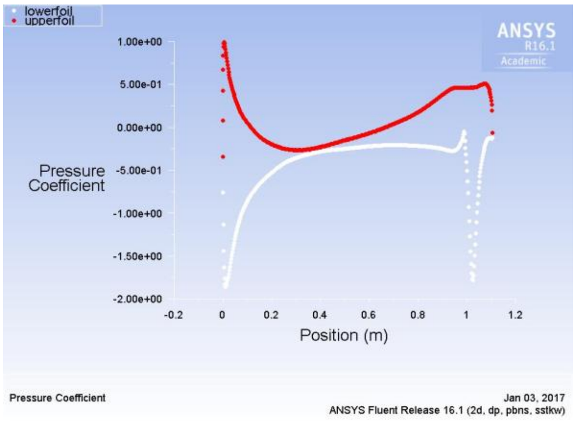

Figure 16. Cp with reflected arc length 12%C

V. CONCLUSIONS

CFD analysis of the NACA4412 is performed using finite volume method in ANSYS (Fluent). At the trailing edge of airfoil, deflector of different length is attached and its effect on airfoil performance is studied. From the study it has been observed that reflector is causes the decrease in lift force and increase in drag force which causes the downforce. With increasing arc length, the lift force is decreases.

REFERENCES

[1] MD. Safayet Hossain, Muhammad Ferdous Raiyan, Mohammed Nasir Uddin Akanda, " A COMPARATIVE FLOW ANALYSIS OF NACA 6409 AND NACA 4412 AEROFOIL", IJRET, 2319-1163 | pISSN: 2321-7308.

[2] Justin L. Smith, Henry Z. Graham, and James E. Smith, "The Validation of an Airfoil in the Ground Effect Regime Using 2-D CFD Analysis", American Institute of Aeronautics and Astronautics (AIAA) 2008-4262.

[3] 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.

[4] Vibin Ninan Titus1, Rohan George Jacob2 & Manikandan S, " CFD Analysis of Variation in Pressure Coefficient for NACA 4412 Aerofoil", Imperial Journal of Interdisciplinary Research (IJIR) Vol-2, Issue-5, 2016 ISSN: 2454-1362.

[5] Karna S. Patel, Saumil B. Patel, Utsav B. Patel, Prof. Ankit P. Ahuja, "CFD Analysis of an Aerofoil", International Journal of Engineering Research, Volume No.3, Issue No.3, pp : 154-158.

[6] S.Kandwal, Dr. S. Singh, "Computational Fluid Dynamics Study of Fluid Flow and Aerodynamic Forces on an Airfoil", IJERT, Vol. 1 Issue 7,September – 2012.