RESEARCH ARTICLE OPEN ACCESS

Analysis of Ground Effect on a Symmetrical Airfoil

Siva V*
*Assistant Professor, Department of Aeronautical Engineering, Sri Ramakrishna Engineering College, Coimbatore- 641 022.

ABSTRACT
A Detailed Study and Computational Fluid Dynamics investigation was conducted to ascertain and highlight the different ways in which ground effect phenomena are present around a symmetrical aerofoil-NACA 0015- when in close proximity to the ground. The trends in force and flow field behaviour were observed at various ground clearances, for different angle of attack. The analysis was carried out by varying the angle of attack from 0^0 to 10^0 and ground clearance of the trailing edge from minimum possible value to one chord length. It was found that high values of pressure coefficient are obtained on the lower surface when the airfoil is close to the ground. This region of high pressure extended almost over the entire lower surface for higher angles of attack. As a result, higher values of lift coefficient are obtained when the airfoil is close to the ground. The flow accelerates over the airfoil due to flow diversion from the lower side and higher mean velocity is observed near the suction peak location. The pressure distribution on the upper surface did not change significantly with ground clearance for higher angles of attack. The lift was found to drop at lower angles of attack at some values of ground clearance due to suction effect on the lower surface as the result of formation of a convergent–divergent passage between the airfoil and the ground plate. The values of drag coefficient were also noted for different ground clearance, which is found to be decreasing as the airfoil is approaching to a closer ground clearance. This ground effect is analyzed using FLUENT 5/6 code.
Keywords – Computational Fluid Dynamics, Gambit, Fluent 5/6 Code

I. INTRODUCTION

In Over the last 20 years, research interest in the various flow phenomena associated with bodies in close proximity to the ground, has been growing. The aerodynamic behaviour of such bodies is distinctly different from that which would be seen about the body if it were placed far from the ground. These distinct flow characteristics are generically referred to as 'ground effect'.

Ground effect is caused by ground interference with airflow patterns around an aircraft when the aircraft is within one wingspan of the surface. If the approach airspeed is too fast, the aircraft will tend to float down the runway, delaying touchdown of the aircraft. Various studies have been undertaken in the past to examine and explain the effect of ground effect on wings or aerofoils, through analytical, numerical and experimental methods. Additionally, various studies have sought to explain ground effect wings or aerofoils, mainly through numerical and experimental means. However this project will demonstrate a simplistic assertion and the influence ground effect on a symmetrical aerofoil using CFD analysis. That is the primary aim of this study.

The main objectives of the present work are:

(a) to study the complete phenomena of ground effect.
(b) to study the pressure distribution over the aerofoil surface at different ground clearances and angles of attack
(c) to measure the lift and drag forces over the surface for different ground clearance and angle of attack.

1.1 **Lift and Drag**

The lift and drag produced by a wing define the performance and general attributes of the craft that it supports. A wing moving through the air produces a resultant force. Lift is defined as the component of the resultant force perpendicular to the velocity vector of the wing. Induced drag is defined as the component of the resultant force parallel to the velocity vector of the wing. There are also other forms of drag, which are collectively referred to as parasite or profile drag, which is the drag created by the friction of the object moving through the air. The total drag of an object moving through the air is the sum of induced drag and parasite drag. Both lift and drag are functions of a number of variables, the density of the air, the velocity of the object through the air and the geometry of the object.

1.2 Downwash

In order to conserve the momentum of the air mass moving around a wing, the flow field before and after the wing is distorted. This phenomenon is known as downwash. As downwash changes the flow around the wing, it affects the relationships of lift and drag to the angle of incidence. Downwash can be represented as a vertical flow component of the free stream velocity and is designated (w).

1.3 Gambit

The gambit software is a pre processor for a fluent code. This is used to design and mesh the working domain. The work domain is generated with the help of our geometry and control volume. The control volume is created by inlet and outlet faces perpendicular to the flow. So the flow entering normal to the inlet face will automatically behave as the free stream air.

1.4 Fluent

The mesh generated in gambit is solved using the appropriate solver, turbulence model and boundary layer. FLUENT is a commercial CFD code that solves the RANS equations on hybrid unstructured grids. It uses a second order upwind discretization based on the SIMPLE pressure velocity coupling and the formation can accommodate incompressible flows.

The core of CFD system FLUENT is numerical iterative solver. it solves equation describing the airflow of the model case by conservation of momentum, mass and energy. Geometric model of the flow must be discretize in the form of computational mesh and equations are solved on this mesh subsequently.

II. INDENTATIONS AND EQUATIONS

The derivatives are written in the following way for convenience:

$C_{L\alpha} = dC_L / d\alpha$ (1)

$C_{M\alpha} = dC_M / d\alpha$ (2)

$C_{Lh} = dC_L / dh$ (3)

$C_{Mh} = dC_M / dh$ (4)

The aerodynamic centre, also known in aircraft, is the point where C_M remains constant with changing pitch angle. It is denoted (aerodynamic) centre in pitch here. Its definition is:

$X_\alpha = C_{M\alpha} / C_{L\alpha}$ (5)

The (aerodynamic) centre in height is the point where C_M remains constant with changing height. It is defined as:

$X_h = C_{Mh} / C_{Lh}$ (6)

III. TABLES AND FIGURES

EDGE NAME	BOUNDART TYPE
INLET	VELOSITY INLET
OUTLET	PRESSURE OUTLET
TOP	WALL
GROUND	WALL
AEROFOIL	WALL

Table.1 Boundary type for each edge

Angle of attack	No of cells	No of elements	No of nodes
0	525847	798112	272265
2.5	238530	360740	122210
5	238430	360590	122160
7.5	238238	360361	121545
10	238432	360593	122161

Table.2 Number of cells and elements for each angle of attack

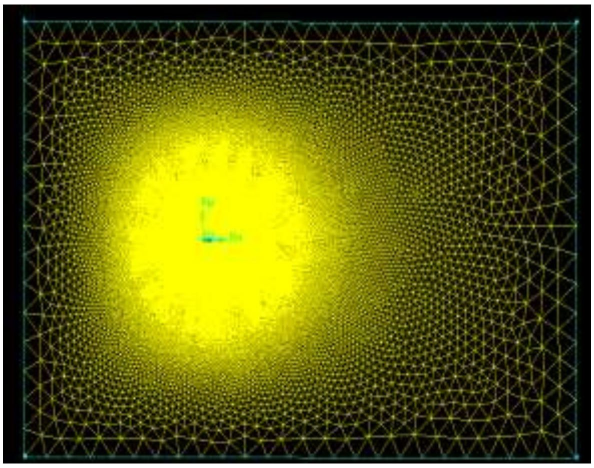

Figure1. Mesh generated using size function

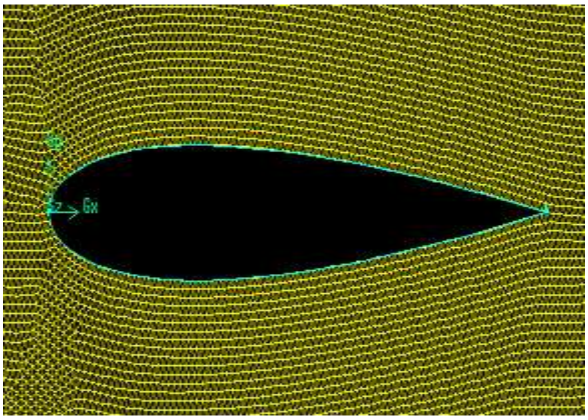

Figure 2.Meshed aerofoil

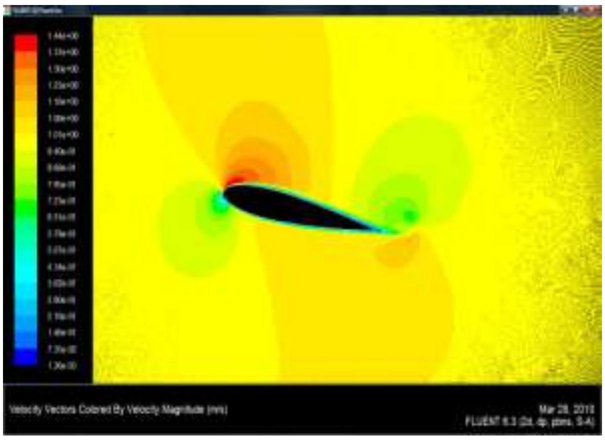

Figure.3 10 degree velocity distribution

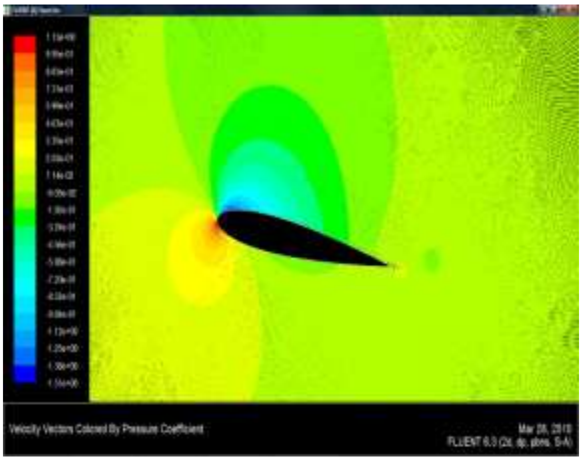

Figure.4 10 degree pressure distribution

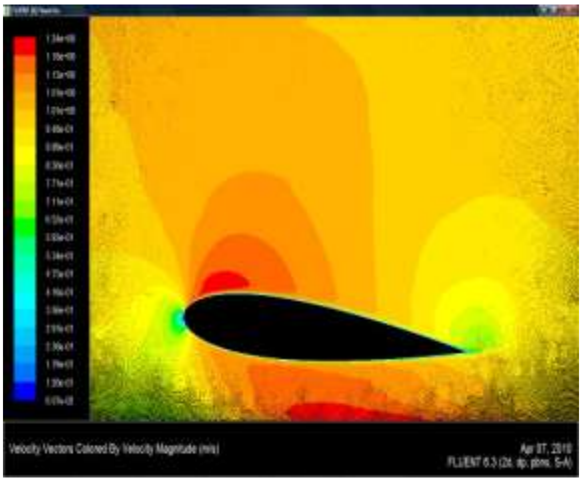

Figure.5 7.5degree.h/c =0.2 velocity distribution

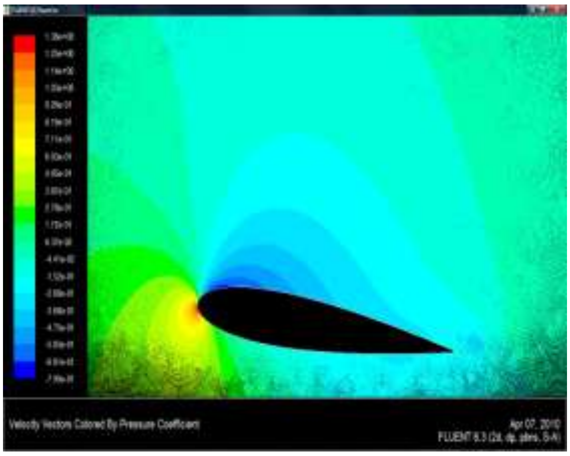

Figure.6 7.5degree,h/c=0.2 pressure distribution

IV. CONCLUTIONS

Result have been presented for computation of flow past a NACA 0015 airfoil using flow governing equations in conjunction with a spalart-allmaras turbulence model for closure. The flow over an airfoil with respect to the ground is analysed using FLUENT and results have been plotted and discussed. Though the results were deviated slightly from the known concept and we could able to justify it still and a detailed analysis have to be carried out to bring a optimum result.

The important conclusions from the present work are:

1. A suction effect is observed on the lower surface at certain ground clearances at angles of attack up to 5^0, due to the formation of a convergent–divergent passage between the airfoil and the ground, causing a local drop in lift force.
2. For very low ground clearances, the lift force was found to be always high, due to higher pressure on the lower surface of the airfoil. At higher angles of attack, high values of pressure coefficient were recorded on the lower surface with the high pressure region extending almost till the trailing edge of the airfoil, which resulted in higher lift force. The pressure distribution on the upper surface did not show significant variation with ground clearance, especially for higher angles of attack; hence, the higher lift force was mainly due to modification of pressure distribution on the lower surface.
3. A reduction in pressure on the suction side was observed at higher angles of attack, causing an adverse pressure gradient on the upper surface, a retarded flow and hence a thick wake region. At very low ground clearances, the airfoil and the ground plate boundary layers were found to merge which resulted in a higher momentum loss and hence a relatively higher drag for higher angles of attack.

REFERENCES

[1]. Ahmed.M.R and Sharma.S.D (2004) 'An investigation on the aerodynamics of a symmetrical airfoil in ground effect' Experimental Thermal and Fluid Science.

[2]. Anderson.J.D Jr, (2001) 'Fundamentals of Aerodynamics, third ed., Mc Graw-Hill' p. 283, 284, 334, 335.

[3]. Barlow.J.B, Ranzenbach. R, (1994) 'Two-dimensional airfoil in ground effect, an experimental and computational study', Society of Automotive Paper No. 94-2509.

[4]. Chang C.H, Chun H.H, (2002) 'Longitudinal stability and dynamic motions of a small passenger WIG craft', Ocean Engineering 29,pp 1145-1162

[5]. Chen C.K., Hsiun C.M, (1996) 'Aerodynamic characteristics of a two dimensional airfoil with ground effect', J. Aircraft 33 (2) p.386–392.

[6]. Ollila R.G., (1980) 'Historical review of WIG vehicles', J. Hydronaut. 14 (3) p. 65–76.

[7]. Robert C. Nelson, (1998) Flight Stability and Automatic Control, 2nd ed., McGraw-Hill.