RESEARCH ARTICLE OPEN ACCESS

Wing in Ground Effect- 2d Analysis Using Ansys

Aananthan M, Aravind VG, Hormese Tharakan, Gouri Shankar Menon
Department of Mechanical Engineering, Scms School of Engineering and Technology,vidya Nagar, Palissery, Karukutty, Ernakulam – 683 582
Corresponding Author : Aananthan M

ABSTRACT
Wing-in-Ground vehicles and aerodynamically assisted boats take advantage of increased lift and reduced drag of wing sections in the ground proximity. At relatively low speeds or heavy payloads of these craft, a flap at the wing trailing edge can be applied to boost the aerodynamic lift. The influence of a flap on the two-dimensional NACA 4412 airfoil in viscous ground-effect flow is numerically investigated in this study. The computational method consists of a steady-state, incompressible, finite volume method utilizing the K-epsilon turbulence model. Grid generation and solution of the Navier-Stokes equations are completed using computer program Fluent. The code is validated against published experimental and numerical results of unbounded flow with a flap, as well as ground-effect motion without a flap. Aerodynamic forces are calculated, and the effects of Reynolds number and ground height are presented for the airfoil. Changes in the flow introduced with the change in ground height are also discussed.
.Keywords - ---

DATE OF SUBMISSION:16-08-2018 DATE OF ACCEPTANCE: 30-08-2018

I. INTRODUCTION

High-speed, high-payload marine and amphibious transportation means benefit from using aerodynamic lift enhanced in ground vicinity. Various types of Wing-in--Ground (WIG) craft, mainly experimental, were constructed and tested in the last century (Rozhdestvensky, 2006). The most effective lift increase and drag reduction are obtained in the so-called extreme-ground flight regimes, roughly corresponding to the wing height over the ground surface being less than the wing chord. A new generation of aerodynamically assisted transport platforms using Power Augmented Ram (PAR) principle has been under development (Gallington, 1987; Kirillovykh and Privalov, 1996; Matveev, 2008). PAR craft operate in the extreme ground effect and rely on both aerodynamic and jet support. The development of complex ground-effect vehicles is hampered by high costs and accuracy issues in conducting experimental studies. Hence, the application of rapidly advancing CFD methods can help the engineering community make a significant progress in the air-assisted fast transportation.

The Wing-in-Ground effect, taking place when an airfoil flies in close proximity to the ground, provides beneficial aerodynamic properties. As the airfoil altitude approaches one wing span, the wingtip vortices become partly blocked by the ground, reducing the lift-induced drag and increasing lift. Further decreasing the ground height to within a small fraction of the wing chord, corresponding to the extreme ground effect, will trap flow underneath the airfoil, providing a high pressure ram effect which significantly increases the aerodynamic lift. It is apparent that the span-based ground effect is a three-dimensional phenomenon. However, the nearly two-dimensional chord-based effect dominates in the extreme ground proximity, when endplates are employed or large-aspect ratio airfoils are utilized.

Many WIG vehicles are designed to skim above water or a relatively flat solid surface. At low speeds or on heavy loaded craft a flap is employed at the trailing edge of the airfoil to strongly augment the airfoil lift. By adjusting the flap position, flow around the airfoil can be controlled, in turn providing control of the aerodynamic properties of the airfoil. In many cases where ground height and the wing attack angle are relatively fixed, a flap can be used to regulate the aerodynamic lift of the vehicle in ground vicinity.

A variety of inviscid numerical methods have been applied in the past for calculating ground-effect flows and even simulating dynamics of WIG craft (Gallington et al., two-dimensional incompressible flows around airfoils in ground effect. A finite volume method employing the k-ε turbulence model was used on an NACA 4412 airfoil by Hsiun and Chen (1996) along with a fixed ground boundary condition. They concluded that a decrease in lift was found in the extreme ground effect due to the boundary layer created between the fixed ground and free stream flow. Different ground boundary conditions on the NACA 4412 airfoil were studied by Barber et al. (1998). They note that the fixed ground condition is unrealistic for WIG craft, and propose the use of a moving ground at the free stream speed. A finite difference scheme and the Baldwin-Lomax turbulence model are used by Chun and Chang (2003) in their investigation of the fixed and moving ground conditions for an NACA 4412 airfoil. Again, they also see a significant difference between a fixed and moving ground. All of the above numerical models, which use the moving ground boundary condition, predict an increase in lift during ground-effect flight, although there is some variation in the magnitude of predicted forces. A few modeling analyses of three-dimensional WIG configurations by viscous solvers were also accomplished (Hirata and Hino, 1997; Wu and Rozhdestvensky, 2001).

Experimental data for wing sections in proximity to fixed ground and moving ground are available in the literature (Hayashi and Endo, 1978). These data correspond to flow of Reynolds number on the order of 105. At these Reynolds numbers the flow is known to be generally laminar on a significant portion of the chord, which makes it difficult to extrapolate such data even for small WIG craft. Kikuchi et al. (2002) present results obtained on a NACA 4412 airfoil towed at a Reynolds number of 8×105. Although the flow is still below the general WIG operating range, practical aerodynamic trends are obtained. They conclude that the ground effect augments the lift in all cases in which geometry does not create the Venturi effect below the airfoil.

Little published research has been found for the use of a flap in the extreme proximity to the ground. Most studied configurations were complicated PAR systems (Huffman and Jackson, 1974; Krause, 1997). Serebrisky and Biachev (1946) tested Clark-Y sections by the method of images in this region and found that use of flap in ground effect improves the aerodynamic efficiency for small angles of attack. Numerous experimental data collected by Abbott and Doenhoff (1959) show that a flap can significantly augment the lift in the free airflow. The flap application has been numerically and experimentally studied by Steinbach and Jacob (1991) in the distant ground effect, with height-tochord ratio from around one quarter to above one. Their numerical technique consisted of a panel method, which was iterated with boundary layer and rear displacement models to account for viscosity and separation. It is concluded that wing systems with excessive flap-slat mechanization are often unfavorable in the distant ground effect as the wing effective camber produces a negative ground effect. Their results also show that as a high-lift airfoil with a flap approaches the ground, the flap efficiency decreases, and the separation point moves further upstream.

However, nearly flat lower airfoil surfaces and moderate flap deflections and attack angles are known to be quite beneficial in ground proximity. The goal of this paper is to investigate favorable trailing-edge flap configurations that improve aerodynamic characteristics of the NACA 4412 wing section in the extreme ground effect. The extreme ground effect region has been chosen as it provides the most beneficial aerodynamic properties for ground-effect vehicles and obtained numerical solutions are generally steady. As a wing with flap moves farther from the ground, unsteady effects begin to take place and an unsteady solution is required. In this work, a numerical study of viscous ground effect flow is completed using the commercial code Fluent 6.3.

II. GROUND EFFECT VEHICLE

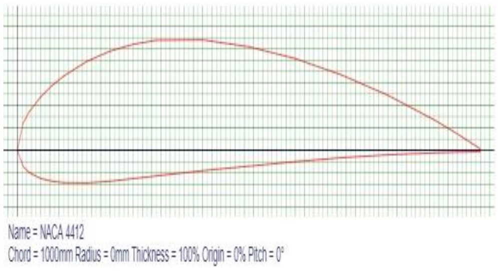
Many vehicles have a design that makes use of the wing in ground effect. Although all airplanes fly through ground effect at some point, craft that do so in a dedicated manner are designed in such a way that their wings are normally unable to take them into flight out of ground effect (free flight). Those that can fly out of ground effect are often capable of only a short distance take-off into free flight. Because of this, these craft are often licensed as ships rather than as aircraft. These specially designed craft may use delta wings, ekranoplan wings or tandem wings.

LUN-CLASS EKRANOPLAN

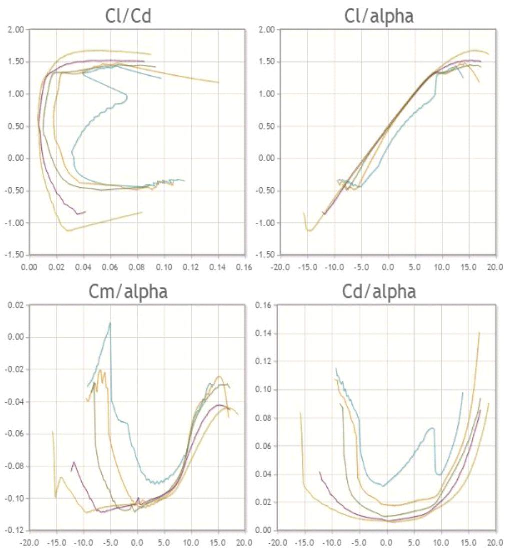
TANDEM-AIRFOIL FLAIRBOAT

1. NACA 4412 AIRFOIL

The NACA airfoils are airfoil shapes for aircraft wings developed by the National Advisory Committee for Aeronautics (NACA). The shape of the NACA airfoils is described using a series of digits following the word "NACA". The parameters in the numerical code can be entered into equations to precisely generate the cross-section of the airfoil and calculate its properties.



III. AIRFOIL CHARACTERISTICS



	Front ($0 \le x < p$)	Back ($p \le x \le 1$)
Camber	$y_c = \frac{M}{P^2}(2Px - x^2)$	$y_c = \frac{M}{(1-P)^2}(1 - 2P + 2Px - x^2)$
Gradient	$\frac{dy_c}{dx} = \frac{2M}{P^2}(P - x)$	$\frac{dy_c}{dx} = \frac{2M}{(1-P)^2}(P - x)$

The thickness distribution is given by the equation:

$$y_t = \frac{T}{0.2}\left(a_0 x^{0.5} + a_1 x + a_2 x^2 + a_3 x^3 + a_4 x^4\right)$$

Where:

$a_0 = 0.2969$ $a_1 = -0.126$ $a_2 = -0.3516$ $a_3 = 0.2843$
$a_4 = -0.1015$ or -0.1036 for a closed trailing edge

- The constants a0 to a4 are for a 20% thick airfoil. The expression T/0.2 adjusts the constants to the required thickness.
- At the trailing edge (x=1) there is a finite thickness of 0.0021 chord width for a 20% airfoil. If a closed trailing edge is required the value of a4 can be adjusted.
- The value of yt is a half thickness and needs to be applied both sides of the camber line.

Using the equations above, for a given value of x it is possible to calculate the camber line position Y c, the gradient of the camber line and the thickness. The position of the upper and lower surface can then be calculated perpendicular to the camber line.

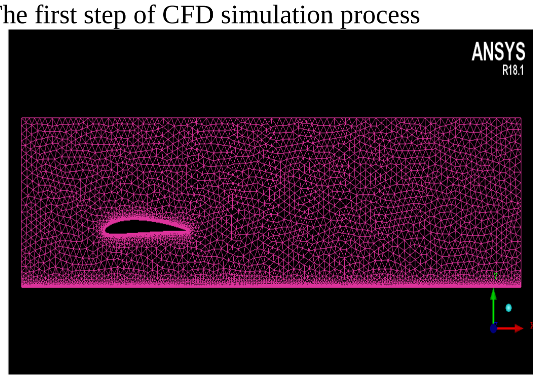
The most obvious way to plot the airfoil is to iterate through equally spaced values of x calculating the upper and lower surface coordinates. While this works, the points are more widely spaced around the leading edge where the curvature is greatest and flat sections can be seen on the plots. To group the points at the ends of the airfoil sections a cosine spacing is used with uniform increments of β

IV. METHODOLOGY

- The geometry and physical bounds of the problem can be designed using computer aided design(CAD).from there data can be suitably processed and fluid domain is extracted
- The volume occupied by the fluid is divided in to discrete cells(THE MESH)/it can be uniform or non-uniform structured or unstructured consisting of a combination of hexahedral, tetrahedral, prismatic, pyramidal or polyhedral elements
- The physical modelling is defined for examples the equations of fluid motion + enthalpy+ radiation + species conservation
- Boundary condition are defined specifying the fluid behaviour and properties of all bounding surfaces a fluid domain. For transient problems initial conditions are also defined.
- The simulation is started and equation are solved iteratively as steady state or transient
- Finally a post processor is used of analysis and visualization of resulting solution

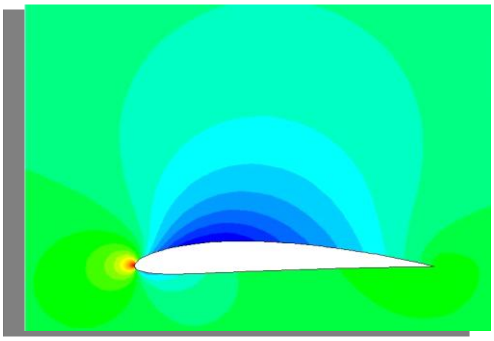
V. MODELING AND SIMULATION

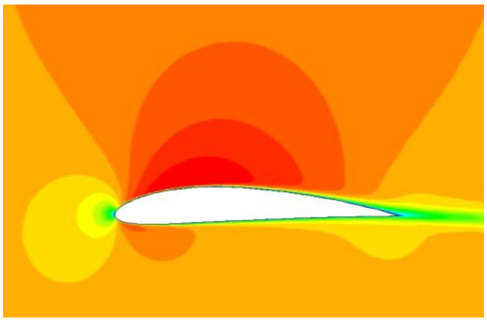
The first step of CFD simulation process



VI. POST-PROCESSING

Figure shows the simulation outcomes of static pressure at angle of attack 0° with k-epsilon model. The pressure on the lower surface of the airfoil was greater than that of the incoming flow stream and as a result it effectively "pushed" the airfoil upward, normal to the incoming flow stream





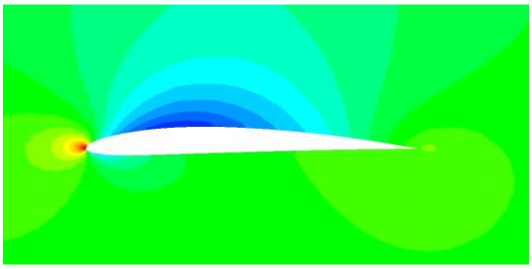
The graph of pressure contour shows a high pressure area below the airfoil due to the ground at close proximity. We can see vortex at the trailing edge of the airfoil which is due to the difference in pressure at the top and bottom of the airfoil. The low pressure area indicate the area of higher velocity. The leading edge experiences area of high pressure and is called the stagnation point.

The area under the airfoil has high pressure air under it which in turn gives a push in the upward direction hence producing lift.

The velocity magnitude contour shows the maximum and minimum velocity on the airfoil. The inlet velocity was given as 30 m/s. The leading edge the velocity is very low and this point is called the stagnation point. The low pressure area indicate the area of higher velocity. The leading edge experiences area of high pressure and is called the stagnation point. The area under the airfoil has high pressure air under it which in turn gives a push in the upward direction hence producing lift.

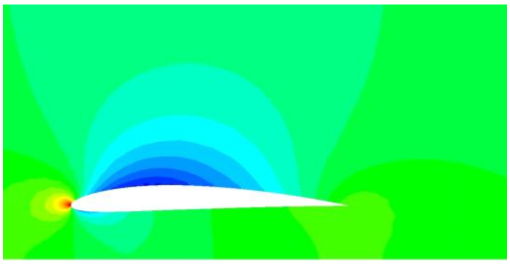
VII. RESULT OF SIMULATION

CONTOUR OF PRESSURE COEFFICIENT FOR VARIOUS H/C VALUES.



h/c=0.75

h/c=0.5

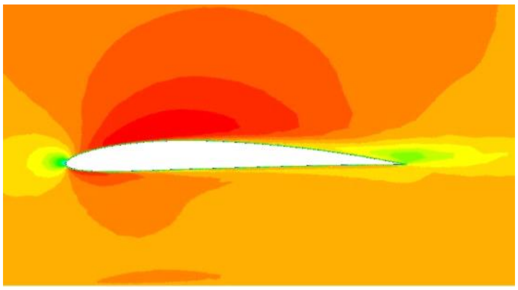


h/c=0.25

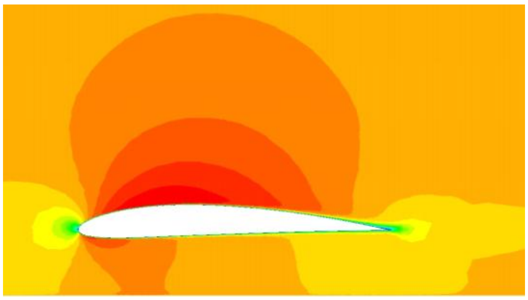
h/c=0.1

CONTOUR OF VELOCITY MAGNITUDE FOR VARIOUS H/C VALUES

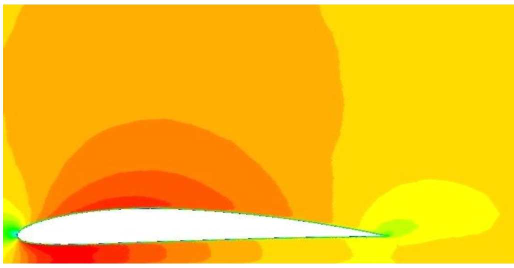
h/c = 0.75



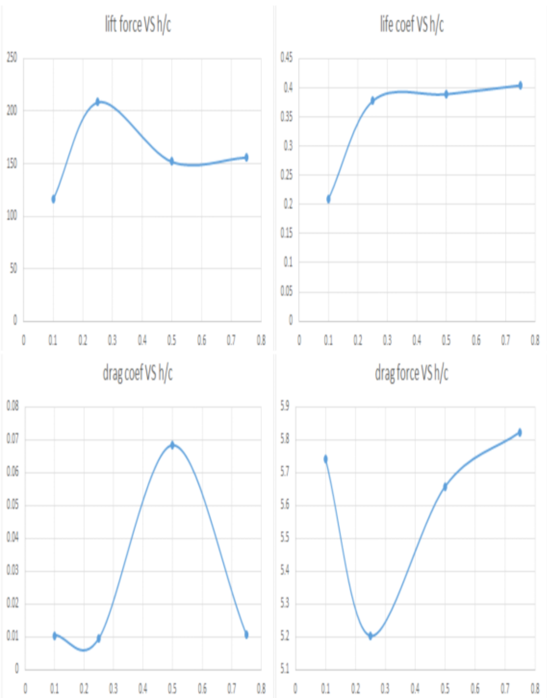
h/c = 0.50



h/c= 0.25



h/c= 0.10



VIII. CONCLUSION

The effect of ground effect can be utilized for higher amount of lift and lower induced drag to create vehicles that carry higher payload with better efficiency. The analysis of the airfoil is completed using Ansys Fluent software. The ground effect characteristics can be further improved by the introduction of flaps to the airfoil and also by changing the angle of attack the airfoil is placed. Further improvement in lift can be made with the reduction in wing tip vortices and by the in-depth analysis of the airfoil. The project imparted us knowledge on wing design, airfoil and modeling software.

ACKNOWLEDGEMENTS

I am greatly indebted to Dr. Praveensal C J., Principal, SSET and Dr.Venu P, Head of Department of Mechanical Engineering, SSET, who whole heartedly granted me the permission to conduct this project.

I would like to thank my project guide Dr. Sheeja Janardhanan, Professor, Department of Mechanical Engineering ,SSET who has given me valuable guidance and support. Also, I would like to thank Mr. Vishnu, Assistant Professor, Department of Mechanical Engineering, SSET who support and instruct me all the way.

I would like to express my sincere gratitude to all the teachers of Mechanical Engineering Department who gave me moral and technical support. I would also like to thank my friends and family members for providing me with necessary resources and support.

REFERENCES

[1]. 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.

[2]. Mr. Monir Chandrala, Prof. Abhishek Choubey, " Prof. Bharat Gupta Aerodynamic Analysis Of Horizontal Axis Wind Turbine Blade", IJERA, Vol. 2, Issue6, November-December 2012.

[3]. Nathan logsdon, "a procedure for numerically analyzing airfoils and Wing sections", The Faculty of the Department of Mechanical & Aerospace Engineering University of Missouri – Columbia, December 2006. www.airfoiltools.com• Airfoil nomenclature.pdf http://ocw.kfupm.edu.sa/ocw_courses/user061/AE33301/Lectu re%20Notes/Flow%20over%20airfoil

[4]. L. Marino. (2004) Induced-drag reduction of wing–wings and wings–ground

configurations. TheAeronauticalJournal 108:1 088, 523-530

[5]. ReadMore: https://arc.aiaa.org/doi/abs/10.2514/2.2711

[6]. M.R. Ahmed, •S.D. Sharma. (2005) An investigation on the aerodynamics of a symmetricalairfoil in ground effect. Experimental Thermal and Fluid Science 29:6, 633-647

[7]. Wei He, Peng Yu, Larry K.B. Li. (2018) Ground effects on the stability of separated flow around a NACA 4415 airfoil at low Reynolds numbers. Aerospace Science and Technology 72, 63-76

[8]. T Lee, A Majeed, B Siddiqui, V Tremblay-Dionne. (2017) Impact of ground proximity on the aerodynamic properties of an unsteady airfoil. Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering 45, 095441001770341

[9]. Ning Deng, Qiulin Qu, Ramesh K. Agarwal. 2017. Numerical Study of Fuselage Aerodynamics of DLR-F6 Wing-Body in Ground Effect. 35th AIAA Applied Aerodynamics Conference.

[10]. Qiulin Qu, Wei Wang, Peiqing Liu, Ramesh K. Agarwal. (2015) Airfoil Aerodynamics in Ground Effect for Wide Range of Angles of Attack. AIAA Journal 53:4, 1048-1061

Aananthan M "Wing in Ground Effect- 2d Analysis Using Ansys 'International Journal of Engineering Research and Applications (IJERA) , vol. 8, no.8, 2018, pp 01-06