External Aerodynamic Study on Wing in Ground Effect (Wig) Marine Vehicle Using Computational Fluid Dynamics (Cfd)

Eferebo Ibietela Sylvanus and Tamunodukobipi Daniel
Marine Engineering Department, Rivers State University, Port Harcourt, Rivers State Nigeria.
Corresponding Author: Eferebo Ibietela Sylvanus

ABSTRACT: *The study of the characteristics performances of a wing in ground effect marine vehicle is carried out in this project; a computational fluid dynamics (CFD) method is employed through numerical simulation to study the behavior of the fluid flow around the 3D WIG craft CAD model. The Navier Stokes Equation, which is based on the Reynolds transport theory, and the K- omega SST turbulence model is solved for the aerodynamic flow around the wing in ground effect (WIG) 3D geometric CAD model. The hex-dominant mesh algorithm was used to discretize a largely enough computational domain to solve for convergence in the numerical simulation, with the goal to obtain coherent and valid simulation run result.*
KEY WORDS: *CFD, Aerodynamics, Wing In Ground Effect, Lift And Drag Coefficients, Advanced Marine Vehicle.*

I.   INTRODUCTION
In recent times, the study of ground effects has maintained its dominance in the technological space of design and development of advanced and fast-moving marine craft, due to the numerous advantages it offers to meet several industrial, commercial and military applications. Before the event of the Captian sea monster, named after Russian, first nationally developed wing in ground effect (WIG) marine vehicle also called Ekranoplane [1]. Several researchers had already understood the benefits of ground effect phenomenon especially during take-off, and landing stages of airplanes in the Aeronautic industries hence continued to explore these benefits such as, in the development of short take-off and landing (STOL) aircraft. They posed with numerous regulatory and design challenges, the extension of these fundamental principles to the marine industry by traditional [2]. Naval architects gave birth to this renewed vision and hope of achieving via design implementation this very fast-moving marine craft. Similar in operations with conventional aircraft but more efficient in comparison with the standard seagoing vessel and equivalent Airplanes, particularly in cases where speed and fuel efficiency become the critical design requirements. Indeed, successful researches in this area have honestly given great promise to the quest and curiosity of the Naval architects on their deep desires to eliminate dominant resistance on the displacement of planning vessel. This study deals with the characteristic's performances of a wing in ground effect marine vehicle, although there are limited, available technical data of existing and operating WIGs in the public domain, to effectively pioneer detailed research and validations in this hybrid and advanced interdisciplinary aspect of naval architectural designs [3]. Computational fluid dynamics (CFD) method is employed through numerical simulation to study the behavior of flow around the 3D model; the compressible flow analysis around the model was carried out in different simulations set up in other to analyze the external aerodynamic influences around the 3D model [4]. The Navies stokes fluid flow equation using the K- omega SST turbulence model solved for the simulation domain discretized with a hex-dominant mesh algorithm.

Background
To the extremity of research, the most efficient shaped body that produces a lift in moving air is the wing — centered on the fact that the comparative factor being efficiency, quantified by L/D is the least for a wing. Amongst other lifting bodies [5]. For an aircraft, the lifting bodies, i.e. moving through the air, produces a differential pressure on the lower surface of the wing bringing about a resultant upward force which

consequently supports the weight of the aircraft. Whereas, for a wig craft, higher static pressure observed, due to the proximity to the boundary.

Over the years there had been several attempts to improve the speed of marine vehicles, especially the ones used mostly for specific application, necessitating the development of several other types of fast marine crafts such as [6], wing-in-surface effect ships (WISES), aerodynamic ground effect craft (AGEC), ram-wing surface effect boats, ram-wing vehicles, air cushion vehicles (ACV), hovercraft, Surface effect ship (SES), wing ships, (Russian for "screen plane" or "low-flying plane") etc. In theoretical hydrodynamics, it is acceptable that an increasing speed will bring about a corresponding increase in hydrodynamic drag forces and resistances [1], in the bid to eliminate or taking to barest.

These drag forces or resistances to achieve high speed and general performances as may also be a requirement in the specific military, commercial, or any **[7]**, other purposeful application, given rise to these several design modifications and innovations of these unique purpose marine craft called WIG. The concept of ground effect and its use to a wing in ground effect marine vessels is a result of several research efforts in the marine transport technology development sector of the global maritime industry [1]. In practical terms, it is the same phenomenon experienced by airplane pilots just before they land along the runway. In such a process, a more significant [5] amount of speed is lost to achieve the complete diminishing of the air cushion. Due to the closeness of the boundary surface to the wing, the flow field around the lifting arm is distorted thereby causing an increase in the lift and a corresponding reduction in the induced drag of the wing. The ground effect even becomes stronger as the craft gets closer to the boundary, the phenomenon reflects the wing tip vortex by building up an air cushion with the lower surface.
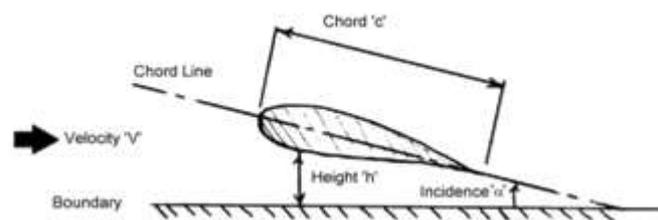

Figure 1: Wing In Ground Effect (Source: Wing In Ground Effect)

The boundary prevents the free expansion of the flow under the wing as applicable to aircraft flying in free stream air out of ground effect. The additional lift is as a result of the rise in static pressure underneath the wing. [8], The total force of the flow field is derived from the static pressure (surface pressure) and dynamic pressure (associated with velocity). As the entire pressure force is constant throughout the flow field, the sum of the static and dynamic pressure will also remain constant [9]. When the flow pushes to the region between the wing and the boundary, the decreasing dynamic pressure, however, converted into a rise in the static pressure which also referred to as 'ram pressure.' The resultant altered pressure distribution [10] causes a net increase in the lift hence the variation to many of the other aerodynamic characteristics of the wing.

Typically, in flying mode, the movement of its wing through air causes a moment whose force arm is about the aerodynamic center [5]. This moment termed the pitching moment consequentially creates a pressure profile on the wing surface which describes the pressure distribution. To maintain equilibrium, the pitching moment characterized by this profile is counteracted by appending another lifting surface in an optimum position, either at the rear (Tailplane) of the aircraft or at the front (canard) of the aircraft. Thus, ground effect alters the pitching moment generated by a wing, thereby moving the aerodynamic center which consequently makes the pitching moment, which is a direct change in the pressure distribution over the lower surface of the lifting wing surface. There is a resultant stabilizing force of a greater magnitude bringing about equilibrium. To ensure stability, a craft in ground effect requires a larger tailplane or canard creating more surface with increased drag which integrally impedes the efficiency of the craft.

Furthermore, another complication enacted by the pitching moment in the ground effect is its change relative to height above the boundary because of alteration in the position of the aerodynamic center relative to the height of the wing above the limit. This complication ensures specific difficulties in design configuration [11]. As a matter of extension, defining the low-speed attribute as well as takeoff and landing speed of the wing is of critical design consideration.

In the case of maximum lift coefficient (Climax), an increase or decrees in this coefficient is dependent on the wing section amongst others. Current researches are based on aerodynamic aids for increasing lift wing design. A factor of similarity between an aircraft and a WIG craft is the aerodynamic behavior. Investigations on the various critical activities such as take-off and landing are expected to be performed by the WIG craft. When

an aircraft stays within one-half of its wingspan close to the ground, theoretically it is believed that it can efficiently fly on one-fifth the power and one-fifth the fuel of a similarly sized aircraft flying at altitude above ground effect [12] and [8]. These were one of the several reasons The WIG craft has greater aerodynamic efficiency (K) and general performance advantages over an equivalent aircraft. So, the relative reduction on the specific fuel consumptions and power is used to derive additional payload.

II. MATERIALS AND METHODS

Overview

The conventional research method was employed to extract data from an ITTC full-scaled model of a planning vessel (SNAME Miscellaneous model resistance data sheets), the Navier-strokes equation with turbulence K-**ω** SST model is solved for the flow around the model. The sufficiently sizeable computational domain is discretized with hex-dominant parametric mesh algorithm with no-slip wall boundary layers' conditions for the compressible flow analysis in ground effects within an altitude h of 0.3m at a constant speed with a varying angle of attack ꭂ. The compressible flow is limited to external aerodynamic performances, such as the pitching moment, Lift and drag influence on the WIG craft. The simulation cases are symmetry in order to save computational time and cost. The CFD simulation is carried out in Simscale commercial software, See more: https://www.simscale.com/. Simscale is a cloud based CFD software platform developed based on OpenForm® solver algorithm.

Preliminary Design of Physical model

The model developed for this research work followed the first two primary phases of engineering design methodology.

However, the design of the WIG vessel is more complicated and tasking than equivalent conventional aircraft and typical displacement or planning Ship. Because there is limited availability of characteristic detailed design data and adequate literature that reflect a balanced knowledge of the various areas of the interdisciplinary field. Although specific rules and guidelines derived from Ship and Aircraft design used as a rule of thumb. The bare hull of the WIG model is a reduced scale (0.12: 1) of a full scaled ITTC planning craft (SNAME Miscellaneous model resistance data sheets no 147) model used. Offsets and ordinates at the design waterline of 0.2m were generated and plotted in inventor 3D CAD modeling software at the scaled value to create the actual hull form. A superstructure profile was modeled and assembled with the hull. NACA 4112 wing profile with coordinates was implemented [5], as rule of thumb for a typical WIG The typical geometric aspect ratio for wig craft ranges from 1 to 3, whereas, those of aircraft are of the order; 5 to 10. The total fuselage length it's 60% of the wingspan (b), in the positioning of the wing on the hull ensuring that coordinates of the highest point of the chamber line coincide
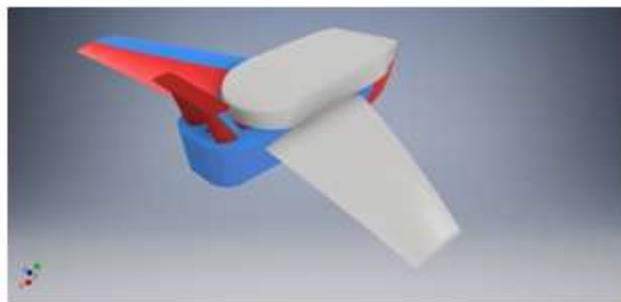

Figure 2: WIG model with NACA 4412

with the center of gravity to provide model stability. Aerodynamic longitudinal and transverse stability criteria at the preliminary stage is checked with some measurable parameters as indicated on the Fig below. Transverse stability at displacement mode followed the traditional Naval Architectural approach with the calculation of the various block form coefficient. Since

analysis is carried out at self-load since vessel meets the requirement for transverse stability, it implies longitudinal stability at displacement state can be neglected.

Navier-Stokes equations governing fluid flow is given as follows;

$$\frac{\partial p}{\partial t} + \nabla(\rho U) = 0 \tag{1}$$

$$\frac{\partial U}{\partial T} + (U.\nabla)U - V.\nabla^2.U = -\frac{\nabla p}{p} \tag{2}$$

$$\frac{\partial pk}{\partial t} + \frac{\partial.U_j.k}{\partial x_j} = G - \beta p\omega_k + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_t.V.t)\frac{\partial k}{\partial t}\right] \tag{3}$$

$$\frac{\partial p\Omega}{\partial t} + \frac{\partial.U_j.\Omega}{\partial x_j} = \frac{r}{V_T}G - \beta p\Omega^2 + \frac{\partial}{\partial x_j}\left[(Y + \sum \Omega V_T)\frac{\partial \Omega}{\partial t}\right] + D \tag{4}$$

Turbulent Viscosity

$$V_t = \frac{P_{a1}k}{\max(a1\omega_1\Omega F_2)} \tag{5}$$

Reynold's Number

$$R_n = \frac{\rho.V.L}{\mu} \equiv \frac{V.L}{\gamma} \tag{6}$$

S/N	Particulars	Dimension (M)
1	Length (L)	1.5
2	Breadth (B)	0.615
3	Depth (D)	0.508
4	Draught (T)	0.2

Table 1: Hull's Principal Particulars

S/N	Data	Dimension (m)
1	Chord length	0.9
2	Aspect ratio	1.8:1
3	Thickness	0.115
4	Span	1.34

TABLE 2: AIRFOIL DATA

Computational Domain

Apart from the first Background Box dimension (33m x 8m x 15.6m) which serves as the simulation domain as shown below in figure 3.4, there were some other internal primitive geometries defined around the craft model for better refinement of the mesh. Minimum and maximum surface refinement levels 3 and 5 respectively was set for the surface around the craft model, an Inflate boundary layers' expansion ratio of 0.3 with minimum thickness of 0.001 and final layer thickness of 0.3 selected likewise, future refinement with an included angle of 150deg with distance level of 0.1m and a refinement level of 5. Region refinement level was defined at level 4 to give a more refined mesh at fillet joints and area where pressure distribution is anticipated to be critical. The setup duplicated and simulation run for other angles of attack of 3, 5 8 and 9deg respectively.
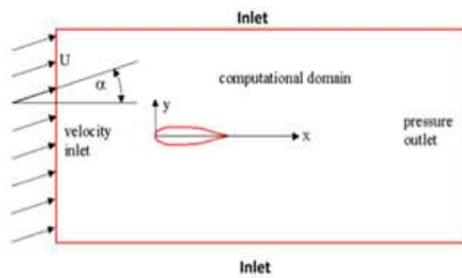
Figure 3: 2D numerical computational domain
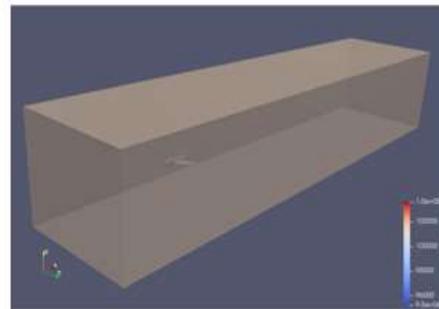
Figure 4: Model View of Computational Domain

Mesh Quality

The Hex-dominant parametric (Only CFD) mesh algorithm was set manually with some default setting on the SimScale CFD platform to discretize the fluid domain. The result of the final run of the mesh as in the figure 35 below was good enough to run the simulation with about 848,562 cells finalizing its parallel run at about 14.3s. In figure 3.6 from the mesh quality matrix in extracted from the mesh result log, the 6 zero column matrix indicates that mesh quality was good and generated without errors.
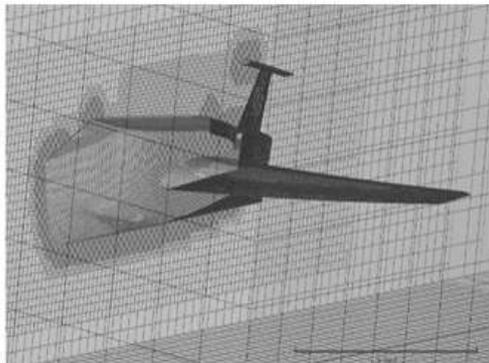
Figure 5: Discretized symmetry   NACA 0012
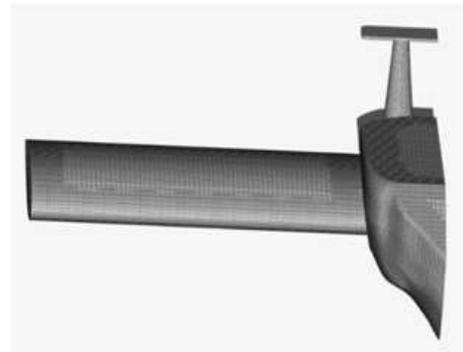
Figure 6: Discretized symmetry for NACA 0012

Grid Generation

The basic hex-dominant mesh algorithm is chosen for the discretization scheme of the model. Hexa-8, 4-dominant, primarily Hex element are made up of 8 components in 3D space while 2D quads have 4 nodes element, the discrete meshed model is made up over 10,000 blocks in the complex analysis. however, with the volume of fluid method, it is stringent to maintain mesh quality in terms of skewness aspect ratio and orthogonality of the mesh, particularly with the complex configuration of the WIG. For Y+ value, generally OpenFoam works with wall function approach when the y+ values are between 30 and 300, therefore the mesh is generated accordingly. However, in full resolution approach the y+ needs to be in ranges between 1 and 5, which would make the mesh size larger, just as the general case, the wall function method was used in this project to appropriate the turbulence effect.

**Table 3: Boundary condition setup presentation**

| Mach number	Inlet velocity	Outlet pressure	Top and side boundaries	Bottom boundary	Symmetry boundary	WIG walls
	32.4	101325	Slip wall velocity	No slip wall velocity	Symmetry	No slip wall velocity

III.   RESULT VALIDATION AND DISCUSSIONS

The validation of results will attempt to discuss results obtained in this research work in comparison with results published in similar work within this field of study.

TABLE 4: TABLE OF PLOTS FOR VARIOUS LIFT AND DRAG COEFFICIENT AT RESPECTIVE AOA

C_L	C_D	C_L/C_D	AOA	C_D	C_M	C_L
0.0422	0.0108	3.907407	0	0.0108	0.0017	0.0285
0.0363	0.0108	3.361111	3	0.0129	-0.035	0.0936
0.0344	0.0108	3.185185	5	0.0166	-0.0465	0.1207
0.0358	0.0111	3.225225	9	0.1144	-0.0232	-0.136
0.0356	-0.0496	-0.71774				
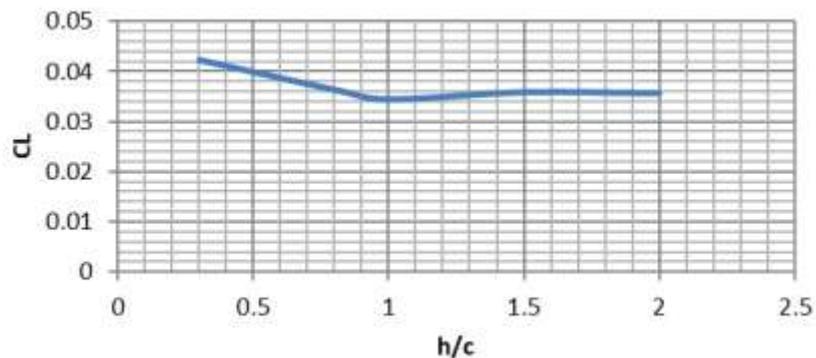
FIGURE 7A: CL VS H/C
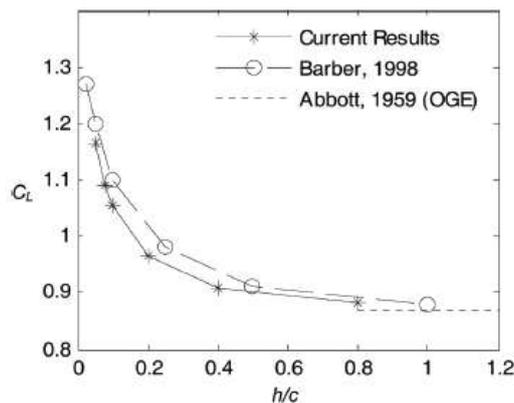
Figure 7b: Validation of C_L vs h/c

Figures 7 (a) and 7 (b) shows the similarity for the NACA 4412 wing profile which depicts some form of analogy, indicating the approximation of the results from the study with those of existing studies. The following figures below show the similarities, though for a portion of the graphs if appropriately inspected. Indicates certain behaviors, if the WIG is to operate beyond the given range within ground effect and as well, out of ground effect. Therefore, it merely means that the (a) part of a figure having h/c at its abscissa illustrates both in the ground and out of ground effect. It should be in mind that the range of in-ground effect approximated between 0 and 0.5m relative to the referenced model chord length. Observations beyond the 0.5 point indicate out of ground effect behaviors of the WIG. For figure 8 below, within range of in ground effect, the coefficient of drag, a variable of the drag force stays constant, even beyond this point. As indicated, if operations have taken beyond a certain point like (1m), the drag force increases, implying beyond the boundary velocity, there remains an optimal value out OGE region to operate, to keep the drag at its barest minimum.
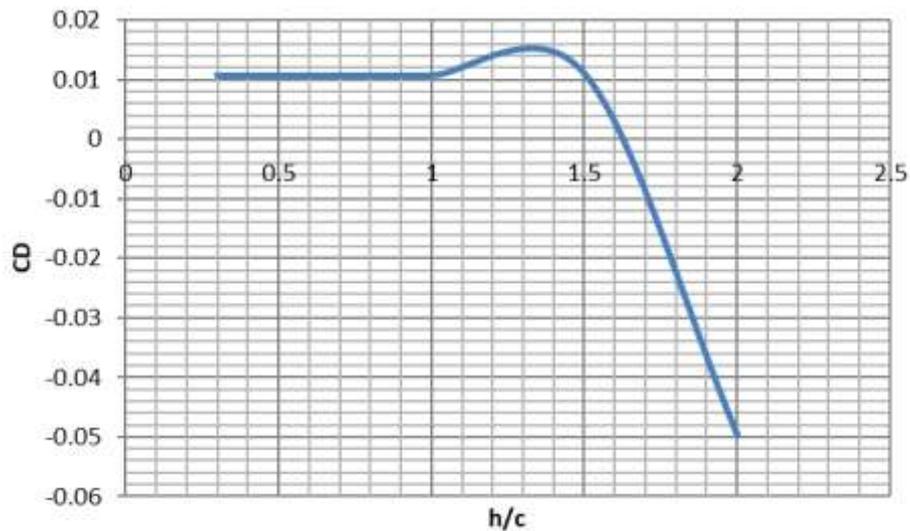
Figure 8: C_D vs h/c

Figure 9 below, the graph indicates the preferred region of in-ground effect (IGE) to be maintained to keep the ratio in favor of the lift and as well, the best operational OGE region to the keep the rate in like manner as it is in the condition of OGE.
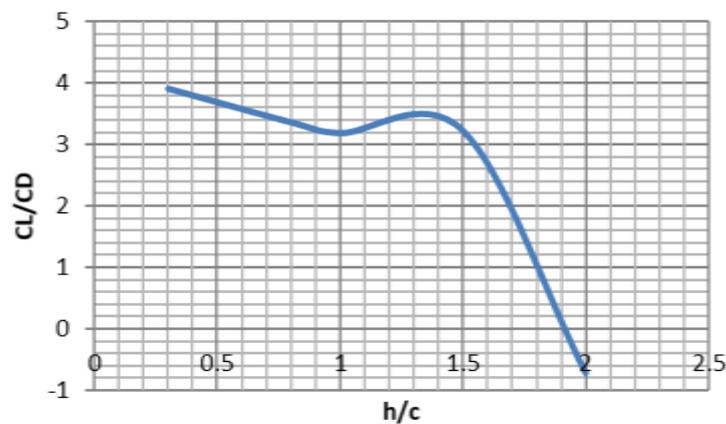

Figure 9: C_L/C_D vs h/c

Closely inspecting figure 9 is a means to validating figure 8 which proves the optimum AOA to operate, to utilize the maximum lift for the given AOA effectively. Therefore, a closer inspection into figure 9 points at AOA of about 4.1 which is within the optimum region to be the most preferred AOA for the study. At this point, the maximum operational lift of the vessel is obtained. At this point also, the drag force stays constant and at its minimum. In comparison to figure 7 (a) the maximum coefficient of lift as indicated is marked at an h/c point of 0.3, for in ground effect.

Provided the operational range kept in focus, a WIG can operate at a favorable AOA to attain even if not the maximum lift possible, at least, it can achieve an approximate lift force desirable for operation. Below is figure 10 which further describes the authenticity of operating the WIG at the marked AOA of 4.1 to be genuinely a satisfactory point of service. Therefore, this point can be taken for standard as an ideal point to be approximated for a given specification of design as this would validate several other factors on which the design of the wing depends and probably the suitable fineness to be attained by the hull.
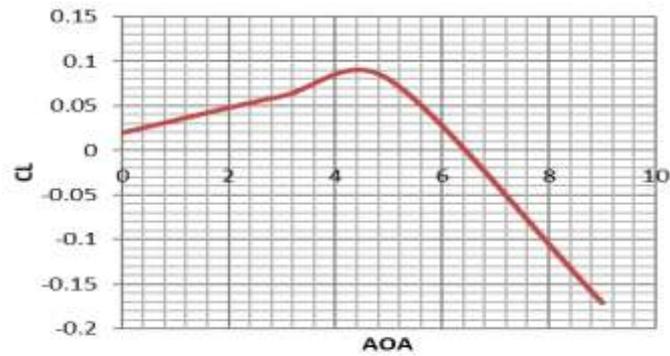
Figure 10a: CL Vs AOA
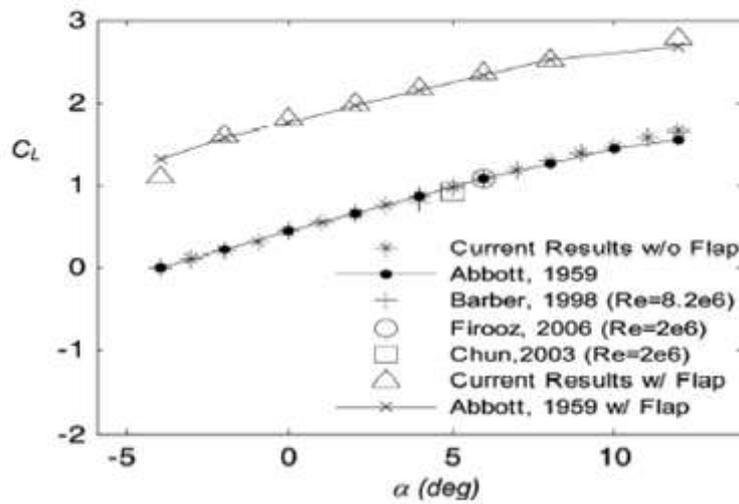
Figure 10a: C_L vs AOA

Designing for a high factor of lift coefficient is a goal to be satisfied during design. However, maintaining stability should not be discarded. for design, the elements to be compromised with should be assured to be favorable to the specific design purpose, to suit the operational mandate.
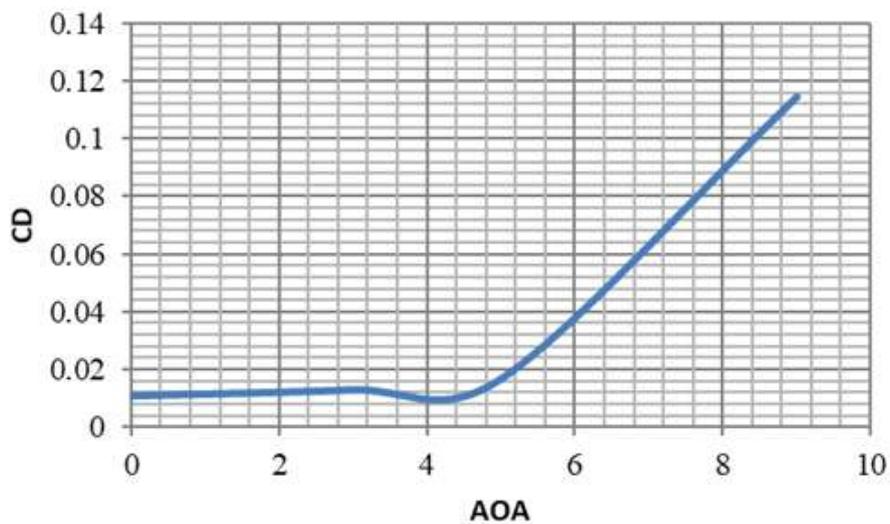

FIGURE 11A: C_D VS AOA
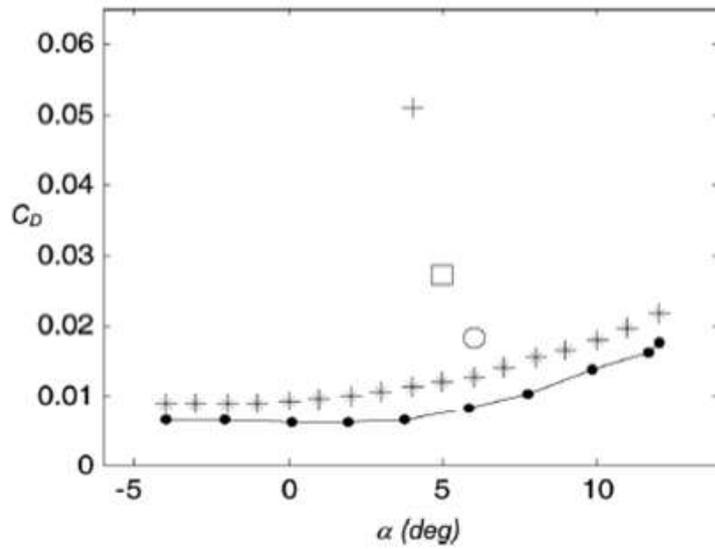
Figure 11b: validating C_D vs AOA

Figure 11 (b) serves as an aid to validate figure 11 (a). Therefore, this explains the similarity between existing data and as well, depicts what relationship tends to ensure within a given region of AOA.
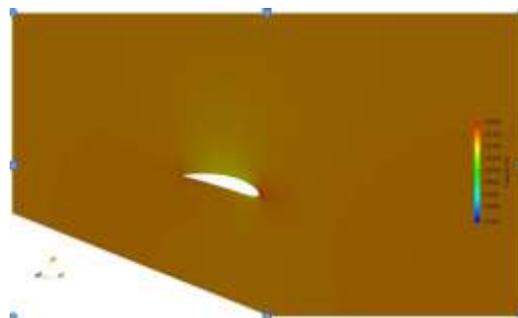
Figure 12: Pressure distribution contours for NACA 4412 Wing Profile
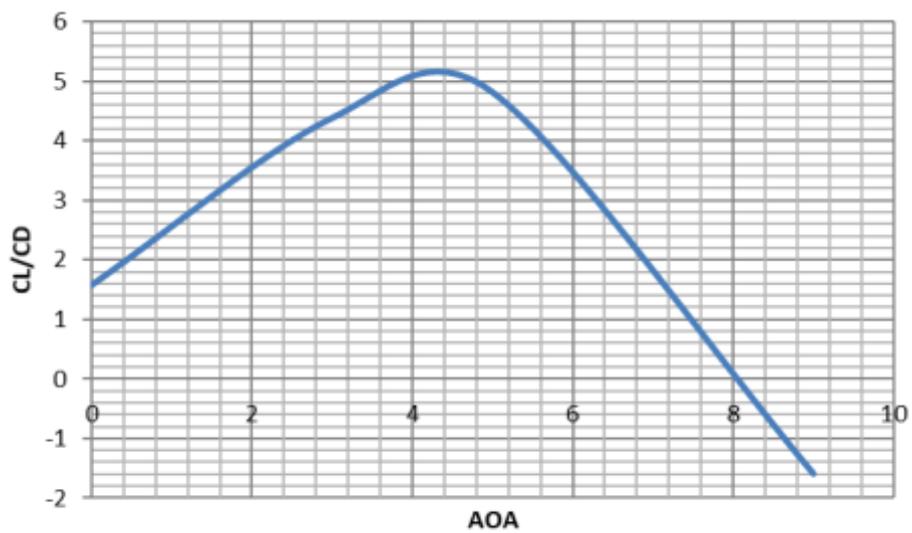
Figure 13: CL/CD vs AOA
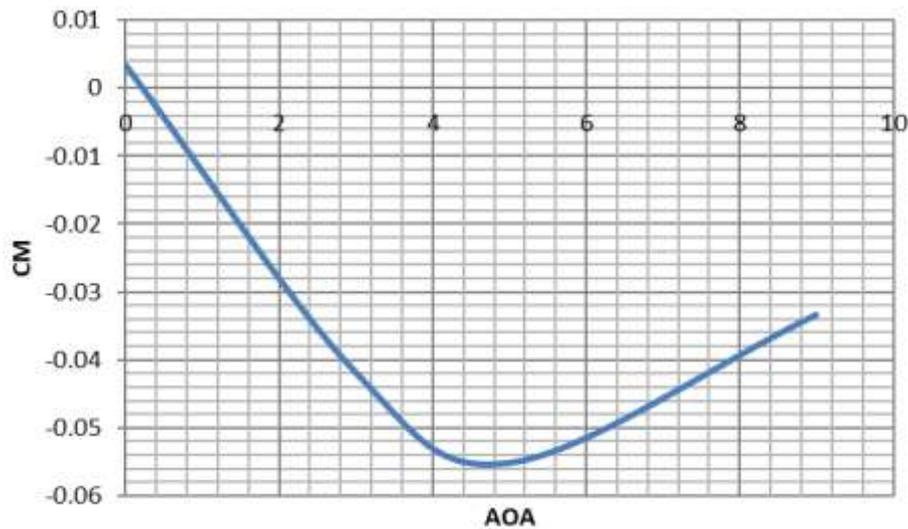
FIGURE 14: C$_M$ VS AOA

In aerodynamics, the pitching moment is that moment (or torque) on an airfoil which is induced by the aerodynamic force on the airfoil provided the point of application of the aerodynamic force is considered to be applied at the aerodynamic center of the airfoil differently from the center of pressure. Generally, for a moving body, a pitching moment is any moment acting on the pitch axis.

The lift on an airfoil, taken as the distributed force can be said to acting at a point called the center of pressure. However, as the AOA varies, for a cambered airfoil, the position of the center of pressure alternates either forward or aft.

An intrinsic property of a cambered airfoil is that; the center of pressure only moves forward or aft, if the lift is assumed to act at a point known as the aerodynamic center, the moment of the lift force changes in proportion to the square of the airspeed. If this moment divided by the dynamic pressure, as well as the area and chord of the airfoil, the following result is known as the pitching moment coefficient. This coefficient varies, but sometimes, gradually, such that only a little over the operating range of the AOA of the airfoil. It is the fusion of the concepts of aerodynamic center and pitching moment coefficient that provides some degree of simplicity in analyzing sure of the flight characteristics or behavior; mainly it's dynamic positioning.
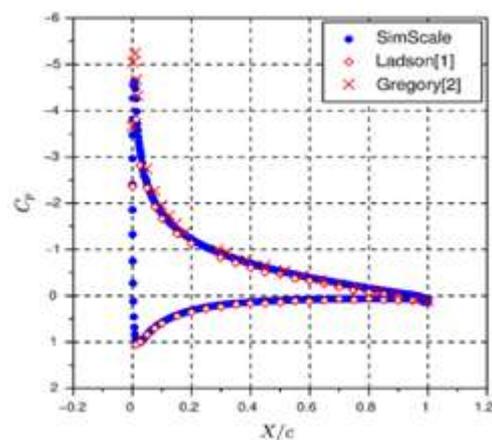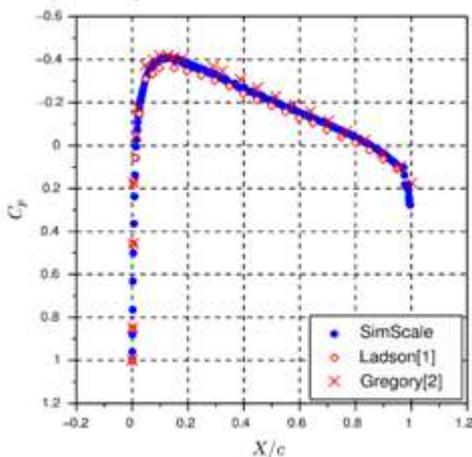

Figure 15: C$_P$ distribution for NACA 0012 OGE at 0° AOA   Figure 16: C$_P$ distribution for NACA 0012 OGE at 10° AOA

In figure 14 above, the more significant portion of the curve lies towards the negative part of the graph indicating a negative moment coefficient for AOA's beyond the point of 0.3. The first need of or indication from a Cm plot is by understanding the craft's stability behavior as it negotiates through the terrain of the dynamic air within the marine environments. Hence, this negative Cm implies that: the wing produces some

magnitude of lift to form a couple which arises for the sake of maintaining longitudinal stability. However, the scenario of negative moment rises basically for cambered wings. Therefore, if the pitching moment coefficient at each point along the chord calculated for each of several values of CL, a critical point is found for which CM is virtually constant and independent of the lift coefficient. This point is the aerodynamic center.

An understanding of how the pitching moment coefficient about a point distance behind the leading edge varies with CL may be used to find the position of the aerodynamic center behind the leading edge and the value of the pitching moment coefficient there. Figure: 15,16 and Figure 17 Below illustrate the comparative CFD validation study of the performance of NACA 0012 in out of ground effect, the results obtained in this research work agrees with the Ladson and Gregory experimental results.
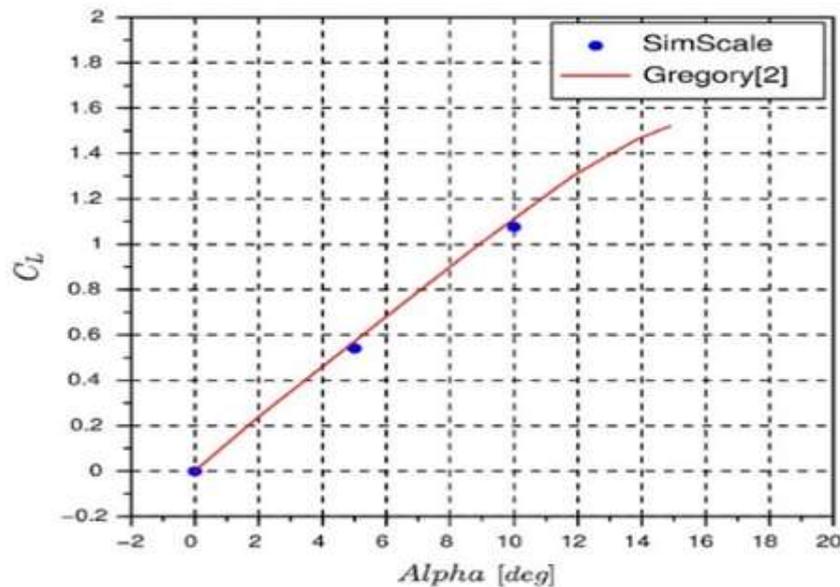
Figure 17: Lift coefficient comparison for NACA 0012 OGE

Aerodynamic properties of different configuration operating at optimal performance have a similar behavioral pattern. WIG craft operating in optimal performance in-ground-effect at acceptable AOA will have an identical measure in establishing its aerodynamic efficiency. Also, a function of performance contributing factors such the L/D and the angle of attack, figure 18 and figure 19 below illustrate the CFD results of NACA 0012 in free stream and out of ground effect. The analysis performed by (NARASIMHAMURTHY, 2007) closely observing the graph and comparing the outcome of this study shows a similar behavioral pattern of the WIG with the operational envelope of the proper angle of attack within ground effect.
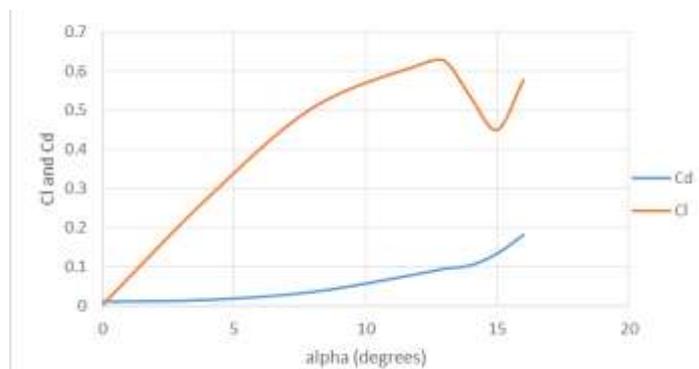
**Figure 18: NACA 0012 lift and drag coefficient vs AOA (Narasimhamurthy, 2007)**
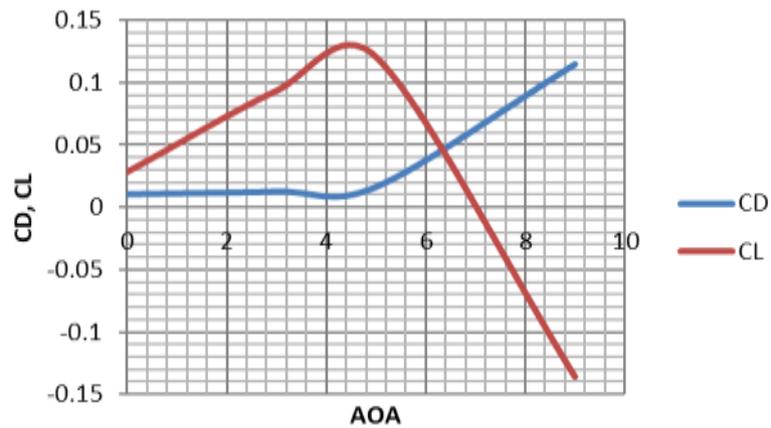
**Figure 19: NACA 4412 in of ground effect; Lift and drag coefficient vs AOA**
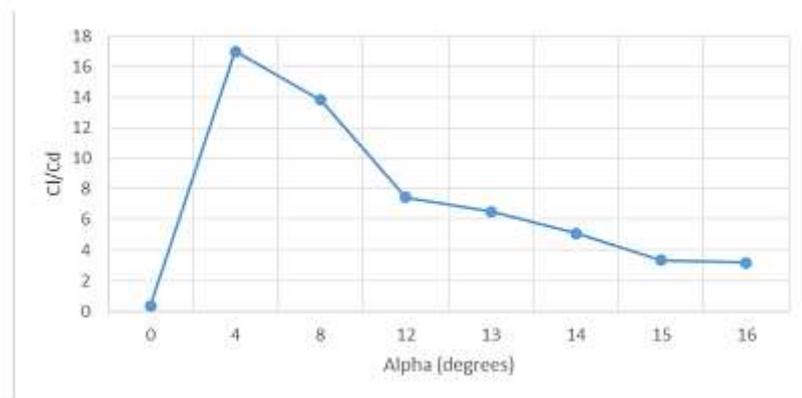
**Figure 20: NACA 0012 CL/CD vs AOA (Narasimhamurthy, 2007)**
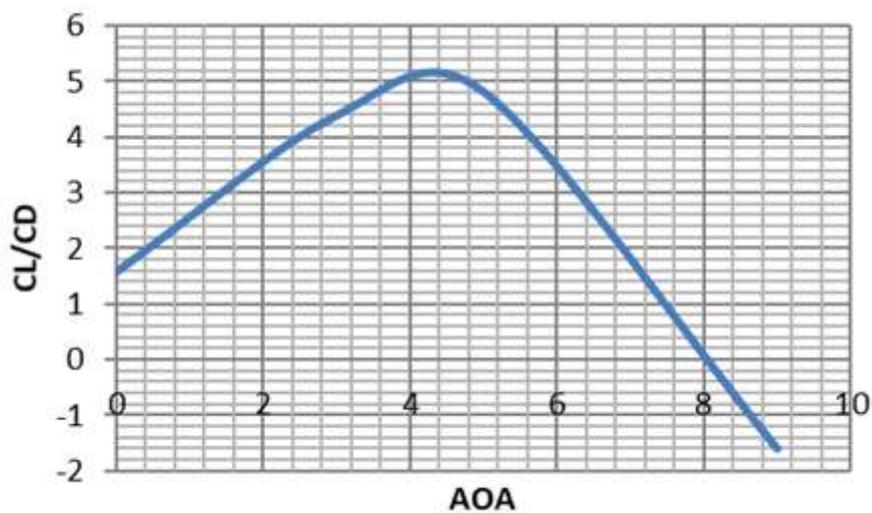
Figure 21: NACA 4412 in ground effect; Ratio of Lift and Drag coefficient vs. AOA

Although there are disturbances associated with the ground effect, from the figures above that the result of this simulation, however, did not deviate from other standard and published result.
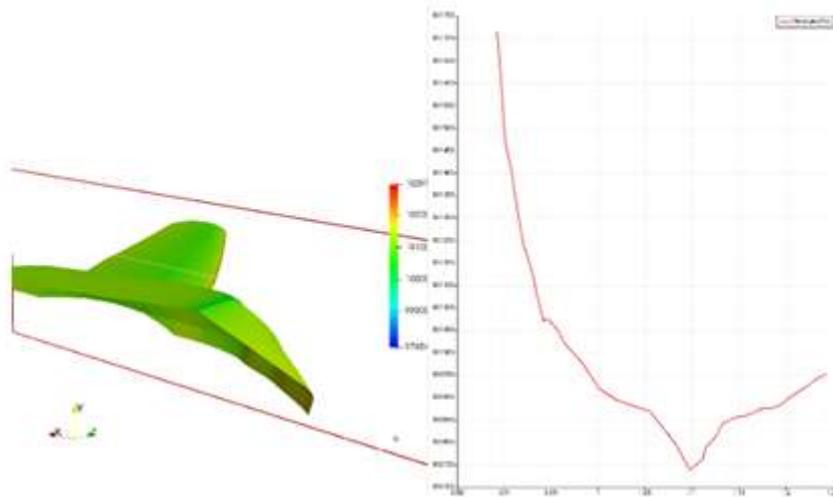
Figure 22: Velocity distribution gradient of NACA 4412 at 0deg AOA
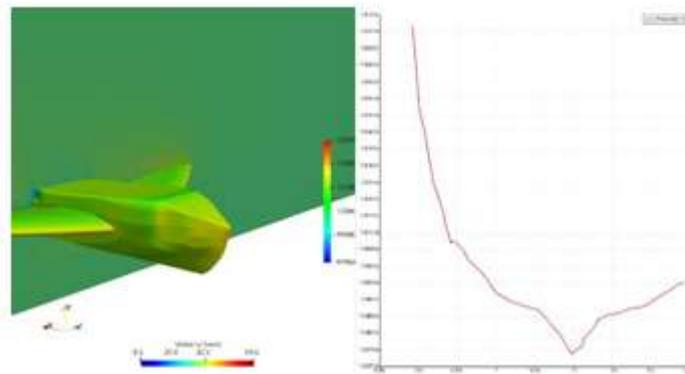
Figure 23: Velocity gradient for NACA 4412 at 0deg AOA

For a given purpose or specification of design, as is expected to be detailed in the mission statement. Figure 23 shows the velocity distribution contours on the craft surface, the velocity in intense at the wing tips as the area with the red gradient indicates the region of high speed on the craft surface operating at 0deg angle of attack.

IV. CONCLUSIONS

The inferences attained from the study have exposed specific hidden facts about the subject. Therefore, it becomes evident as to what variables determine the ideal operation of the craft, with the varying conditions to evaluate as well as possible factors that could be altered to operate optimally at whatever state of service. In totality, the aim and objectives of the study been achieved by the statistical data generated as well as sequences of validation done in line with existing and standard results.

Increasing the angle of attack will certainly increase the L/D value implying better lift at reduced drag forces, however at very strong ground effect zone the angle of attack has an optimal value to which it's not expected to exceed in other not to produce negative lift.

NACA 4412 profile for ground effect is a more efficient and better in consideration and comparison with NACA 0012 profile.

For a given craft it is obvious that the lift to drag ratio is higher at ground effect hence sustaining a better aerodynamic efficiency compared to equivalent craft operating in free stream.

Conclusively, the study has dealt with two significant profiles for analysis and supports a given pattern for a specified mode or need of an operation. Its recommended that relative to a given design task, the hull form should typically have a property as that used for the study or a much more exceptional hull to have a much finer hull form co-efficient and consequently, better performance. Assures better hydrostatic, aerodynamics and hydrodynamic behavior, such that the demands of design approximated. Hence, the results here used with adherence to the prescribed methodology to acquire the reached results effectively. Therefore, to meet up with

certain standards of design, it is therefore required that the sequence of design be also approached with adherence to the regulatory rules as opined by the regulatory bodies concerned for the design and construction of WIGs in that jurisdiction.

Mesh independent study should be carried out to ensure the validity of the grid size to be selected to achieve better CFD result.

The hydrodynamic analysis should be carried out for a wide range of heel angle, Froude number and at different design draught to accurately reveal the variable hydrodynamic characteristics of the planning hull (WIG craft), particularly the resistance performance.

ACKNOWLEDGEMENT

REFERENCES

[1]. Liang Yun, A. B. (2010). WIG Craft and Ekranoplan - Ground Effect Craft Technology. Springer New York Dordrecht Heidelberg London: Springer.
[2]. Paek, C. S. (2006). The viability of commercializing Wing-In-Ground (WIG) craft in connection with technical economic and safety aspects followed by IMO legislation.World Maritime University Dissertations, 120.
[3]. Wong, C. T. (2018). CFD Simulation of a Wing-In-Ground-Effect UAV. International Conference on Aerospace and Mechanical Engineering (AeroMech17) (p. 9). The Hong Kong Polytechnic University, Hong Kong: IOP Publishing.
[4]. Phan Quoc Thien, N. K. (2015). Numerical simulation of floating airboat: Estimation of hydrodynamic forces. International Journal of Mechanical Engineering and Applications, 6.
[5]. Abdul, G. (September 2015). Win In Ground Effect: Modelling and Control. Middle East Technical University.
[6]. N.Moore, P. P. (January 2002). An investigation into wing in ground effect airfoil geometry. Researchgate, 28.
[7]. B.C. Khoo, a. H. (2016). The Hydrodynamics of the WIG (Wing-In-Ground) Effect Craft. Singapore.:   Science direct.
[8]. Alexander Nebylov, D. D. (2014). Flight Automatic Control Systems For The Wing-In-Ground Effect Craft Buchon-1. State University Of Aerospace Instrumentation,, 6.
[9]. M A U Amir, A. M. (2016). Computational analysis of aerodynamic characteristics for wing in ground effect craft in lateral stability. IOP Conference Series: Materials Science and Engineering (p. 10). IOP Publishing.
[10]. D. James, M. A. (2005). Aerodynamically Alleviated Marine Vehicles (AAMV): a review of the main challenges and hydrodynamic aspects. Ocean Systems Test Laboratory, Cranfield University, Cranfield, UK.
[11]. Rahimuddin, A. M. (2014). Stability Analysis of a Wing in Ground Effect Craft. International ShipStability Workshop (ISSW), 8.
[12]. Wolf, W. d. (2002). Aerodynamic investigations on a wing in ground effect. National Aerospace Laboratory, 16.
[13]. Kemal, K. a. (2016). Mathematical Model for Takeoff Simulation of a Wing in Proximity to the Ground. Istanbul Turkey: J. Marine
[14]. MBA, G. K. (November 1997). Market Focused Design Strategy Wing-In-Ground Effect Vehicles; Viable Transport System or Flight of Fancy? International Wing-In-Ground Effect Conference, The Royal Institution of Naval Architects, London., (p. 16)