# Takeoff analysis of amphibious aircraft with implementation of a hydrofoil

Arjit Seth<sup>1)</sup>, and Rhea P. Liem<sup>2)</sup>

<sup>1)</sup> Department of Aeronautical Engineering, Manipal Institute of Technology, Manipal Academy of Higher Education, Karnataka, India <sup>2)</sup> Department of Mechanical and Aerospace Engineering, The Hong Kong University of Science and Technology, Clear Water Bay, Hong Kong <sup>1)</sup> arjit.seth @learner.manipal.edu <sup>2)</sup> rpliem @ust.hk

# ABSTRACT

This paper analyzes the effects of a hydrofoil in takeoff performance improvements of an amphibious aircraft. The physical framework incorporates hydrodynamic effects such as lift, drag and cavitation effects from the hydrofoil, computed using high-fidelity cavitation and turbulence models in ANSYS Fluent along with a preliminary 2D analysis. The takeoff performance is analyzed using an in-house code developed in Python, using the Froude resistance drag model and preliminary thrust estimation from a conceptual design based on historical data. Stability is considered by calculation of required trimming forces from the empennage during the takeoff run.

## 1. INTRODUCTION

Amphibious aircraft are playing an increasingly important role in the aircraft industry. The AG-600 developed in China is effectively the largest amphibious aircraft developed in terms of functionality and passenger transport. Some modern amphibious aircraft exist in general aviation such as the Icon A5 for leisure flying purposes. Amphibious aircraft can also play an important role in payload transport over short ranges, and fuel efficiency is always a key consideration in the design and development. Most of the research conducted in the 1940-50s was focused on empirical methods (Hugli and Axt 1952), with several restrictions in analysis that did not allow for generalizations in size.

Three key non-dimensional parameters in water analysis are the Reynolds number, Froude number and Weber number:

<sup>&</sup>lt;sup>1)</sup> Undergraduate Student

<sup>&</sup>lt;sup>2)</sup> Assistant Professor, Ph.D

The 2018 Structures Congress (Structures18)

Songdo Convensia, Incheon, Korea, August 27 - 31, 2018

$$Re = \frac{\rho VL}{\mu}, \quad Fr = \frac{V}{\sqrt{gL}}, \quad We = \frac{\rho V^2 L}{\sigma}$$

The Weber number does not scale accordingly with Reynolds and Froude numbers because it goes as the square of the velocity, so the benefits of non-dimensional analysis are somewhat lost when attempting to size a water-based component of a transport vehicle based on a scale model. The case worsens with the Froude number going as the reciprocal of the square root of the length against the Reynolds numbers' proportional relationship to length. The mitigating strategy is to use the same Froude number in the model tests and to adjust for different Reynolds numbers when scaling. Some errors exist in water spray, wave pattern and foaming predictions due to the difference in Weber numbers, but these are negligible in resistance prediction of scaled-up hulls (Larsson and Raven 2010). Hydrodynamic forces in hull design are non-dimensionalized by division with  $\rho_W g B^3$ , where *B* is the width of the hull, as opposed to the dynamic pressure and area; this is usually because the hull lengths are fixed and the widths are varied for design analyses. Archimedes' principle also justifies the inclusion of gravitational acceleration as a term to account for buoyancy forces.

With the advent of greater computational power in the modern era, the use of computational fluid dynamics (CFD) has surged greatly in aerodynamic analyses and now plays a key factor in aircraft design analysis and optimization. Hydrodynamic analyses have also been performed on ship hulls using CFD to optimize their design and performance (Frisk and Tegehall 2015). Computational models for takeoff analysis of amphibious aircraft have a large scope for development, and provide initiatives to optimize CFD analyses of air-water interface-based computational models. One area with potential for research and development is the implementation of hydrofoils in amphibious aircraft.

#### **Hydrofoils**

A hydrofoil is defined as a lifting surface that travels through water. Hydrofoils have been researched and implemented in water-based crafts since the late 1800s. It has been extensively researched between the 1930s to the 1950s to improve performance of marine vehicles by allowing a ship's hull to reach a hydroplaning stage more quickly, thus reducing motor effort in a shorter time-frame by reducing the overall hull drag. The boats competing in America's Cup have also implemented hydrofoils in their designs for the same purpose.

The LISA AKOYA<sup>1</sup> is currently the only operational amphibious aircraft in the world known to implement this technology in its design. However, it is only a two-seater aircraft designed for leisure flight, and no publicly available data exists on the benefits of hydrofoil performance for this aircraft. This technology is expected to reduce takeoff distance with the additional lifting forces provided in aircraft designed for civil aviation. One of the goals of the implementation is to provide a "riding" surface for an amphibious aircraft while it is in

<sup>&</sup>lt;sup>1</sup> http://lisa-airplanes.com/en/light-amphibious-aircraft-akoya/

a hydroplaning stage to reduce hull drag by minimizing water contact, so the aircraft is effectively surfing on the hydrofoil. However, there are extensive complications with analyses of hydrofoils. The major issues are effects of cavitation and ventilation.

Cavitation is a complex, turbulent phenomenon that takes place in water when the local pressure is below the saturated vapour pressure, so bubble and vapour formation take place along the lifting surface. This is known to cause "cavitation damage" in the form of corrosion of rotor blades of boat motors. In the case of aircraft, the relevant disadvantageous effect should be the large increase in drag generated by the hydrofoil due to cavitation, and possibly some form of cavitation damage along the hull. The scope of the current work is restricted to analysis of the former. Cavitation is inevitable at high speeds. There are three classifications for cavitation of hydrofoils (Vagianos and Thurston 1952):

1. Subcavitating: Fully attached flow over the hydrofoil.

2. Cavitating: Flow separation takes place at some transition point over the upper surface of the hydrofoil.

3. Supercavitating: The entire upper surface of the hydrofoil undergoes separated flow. Ventilation is a phenomenon that takes place when a lifting surface (such as a hydrofoil) pierces the water and air gets sucked down this surface. This has negative effects on performance such as reduction of lift curve slope and losses of lift up to 75% as compared to hydrofoils at 'infinite' depth (Vagianos and Thurston 1952).

A preliminary hand-calculation shows that the Reynolds number in water is 16 times greater than the Reynolds number in air for the same speed and Reynolds length. Lift and drag forces in water are approximately 815 times ( $\rho_W/\rho_A \approx 815$  at sea level conditions) greater for the airfoils with the same profile, dimensions and non-dimensional coefficients travelling at a given speed in regimes of completely attached flow (low angles of attack). The Froude number also becomes an unreliable measure as the non-dimensionalization relies on the length of the lifting/buoying surface. The possibility of longitudinal oscillatory motion of the aircraft exists, and the hydrofoil may exit the water and pierce it at different times. This results in inconsistent measurements, therefore variation of forces over time is analyzed as well in this paper.

# 2. RELATED WORK

Efficient water takeoff performance analyses of amphibious aircraft have been performed which implement a decoupled analysis method that greatly reduces computational time (Qiu and Song 2012). This strategy is optimal in aircraft analysis using VOF methods for air-water interfaces. Research on performance of hydrofoil-craft such as the influence of foil size and angle on take-off speed have been conducted (Latorre and Teerasin 1992). There is substantial literature on the development of hydrofoils with experimental tests by various researchers since the 1940s. Design of wing sections for hydrofoils in the development of hydrofoil craft have been extensively conducted (Eppler and Shen 1979). In the past decade, an increased focus on numerical solutions of flows around hydrofoils

using CFD has taken place. The literature has become sophisticated to the extent of hydrodynamic shape optimization (Garg *et al* 2015a) and multipoint hydrostructural optimization of 3-D hydrofoils with subcavitating regime constraints (Garg *et al* 2017). However, the literature of hydrofoil applications in seaplanes and amphibious aircraft is highly sparse.

The Thurston Aircraft Corporation conducted research on hydrofoil seaplane design in the 1970s with extensive focus on cavitation, suggesting that hydrofoils designed for supercavitating flows are superior to those designed for subcavitating flows in high-speed flow regimes where cavitation cannot be avoided (Vagianos and Thurston 1952). Computational frameworks of takeoff analyses regarding amphibious aircraft with hydrofoils is highly sparse and cannot be easily found, if they are not non-existent so far.

## 3. COMPUTATIONAL ANALYSIS MODELS

## Baseline Model

The aircraft has been sized around a DHC-6 Twin Otter, with the fuselage modified to include a low-drag, planing-tail, flying-boat hull from the NACA Technical Note 2481 (Suydam 1952). It is assumed to be in still air and operating in a calm and smooth body of water<sup>2</sup> at sea level for the sizing procedures and takeoff setup. The center of gravity (CG) of the aircraft has been initially fixed at 33% of the mean aerodynamic chord of the wing from the leading edge. It is also assumed that all forces generated from lifting surfaces act at the aerodynamic centers of the respective components, which are assumed to be located at approximately 25% of their respective mean aerodynamic chords from the leading edge.

| Parameter          | Specification | Description            |
|--------------------|---------------|------------------------|
| Aircraft Weight    | 5670 kg       | DHC-6's MTOW           |
| Wingspan           | 19.8 m        | DHC-6's wingspan       |
| Aspect Ratio       | 10            | DHC-6's aspect ratio   |
| Wing Configuration | Top wing      | DHC-6's configuration  |
| Wing Dihedral      | 3°            | DHC-6's dihedral angle |
| Hull Configuration | Planing hull  | NACA TN-2481           |

Table 1: Aircraft Configuration and Dimensions

<sup>&</sup>lt;sup>2</sup> Level 0, World Meteorological Organization: http://www.wmo.int/pages/prog/amp/mmop/faq.html



Fig 1: Aircraft Technical Specifications

## Aerodynamic Calculation for Takeoff

The aerodynamic effects that will be calculated from this aircraft are the lift and drag from the wing. The required aerodynamic forces on the empennage are calculated from the moment calculation results to determine the stability and required trim angle of the elevator during takeoff including the hydrofoil within physically viable limits.

## 2D Setup

A structured mesh is generated around both lifting surfaces to obtain accurate results. The first layer thickness of the boundary layer mesh is  $\Delta y_1 \approx 1 \times 10^{-5}$  to obtain  $y^+ < 1$  with given computational limitations, with the k- $\omega$  SST turbulence model. The mesh is generated using ANSYS ICEM CFD with approximately 50,000 elements for a preliminary analysis.



Fig 2: 2D Aerodynamic Mesh

A study has shown that the ground effect between the wing and waterline height is negligible to the order of less than 1% for a height of 3 m compared to freestream values of lift force (Qiu and Song 2012). Since the height of the aircraft wing is approximately 3 m, ground effects can be safely ignored for a preliminary study.

Hydrodynamic Calculation for Takeoff Hull

A major factor in hull design is keeping the trimming moment close to 0 at low speeds, when the elevator is ineffective in providing trim. There are primarily two types of hull drag: frictional resistance and wave resistance, non-dimensionalized via the Reynolds and Froude numbers respectively. Resistance due to vapor pressure is also important in detailed analyses, but its contribution is no more than 5% (Qiu and Song 2012). It is conventional to perform experiments centered around the Froude number, as mentioned in the introduction, to accurately model wave drags. The International Towing Tank Conference has provided a detailed method for calculating full-scale frictional resistance

based on towing tank results of a model called the ITTC-78 method, which relies on the ITTC-57 frictional correlation formula:

$$C_f = \frac{0.075}{[\log_{10}(Re_{hull}) - 2]^2}$$

However, a full-scale analysis of hull resistance is out of the scope of this paper, since only the difference between the takeoff distance of the aircraft with and without the hydrofoil is under investigation. There are also disadvantages with a viscous implementation such as excessive computational time with a viscous flow analysis, which would not contribute to the development of the research scope. A method based on the load coefficient and the Froude resistance drag model (Gudmundsson 2014) is adopted instead to compute the hull drag, in line with the NACA TN-2481 research.

The load factor  $C_{\Delta}$  is defined as:

$$C_{\Delta} = \frac{\Delta}{\rho_W g B^3}, \quad \Delta = W - \sum_c L_c$$

Let  $C_{\Delta_0}$  denote the maximum submerged volume (i.e. when the aircraft is stationery), also known as the 'gross load coefficient':

$$C_{\Delta_0} = \frac{W}{\rho_W g B^3}$$

A variation of the Froude number, the velocity coefficient is defined as:

$$C_V = \frac{V}{\sqrt{gB}}$$

This definition was motivated by the comparison of hull shapes and their dimensions. The NACA TN-2481 planing hull provides empirical data on trim variation and load coefficient against this quantity and the resistance coefficient, defined as:

$$C_R = \frac{R}{\rho_W g B^3}$$

The empirical data shown of  $C_R$  against  $C_V$  is curve-fitted to a series of sine functions.

$$C_{R} = f(C_{V}) = \sum_{i=1}^{4} a_{i} \sin(b_{i}C_{V} + c_{i})$$

The hull trim angle is fitted to a hyperbolic tangent curve.

$$\alpha_{\rm trim} = m + \tanh(nC_V + r)$$

where  $a_i$ ,  $b_i$ ,  $c_i$ , m, n and r are constants. These will provide the resistance and trim angle produced by the hull for any required inputs in the takeoff calculator.





The resistance is assumed to be dependent on the volume of the hull submerged, which is represented as  $C_{\Delta}/C_{\Delta_0}$ . When  $C_{\Delta} \approx 0$ , the aircraft may not have achieved enough lift for takeoff, and could be hydroplaning instead. The Froude drag resistance is introduced at this stage as a correction term with the model:

$$R_{\rm Froude} = f S_{\rm wet,hull} V^n$$

where  $f \approx 0.012$  for a smooth surface and n = 2 according to literature (Gudmundsson 2014). The wetted area of the hull in the hydroplaning state is dependent on the position of the hydrofoil but is assumed to be 3  $m^2$  for this analysis.

#### Hydrofoil

A surface-piercing hydrofoil configuration with a YS-920 foil shape is selected based on availability of empirical data (Eppler and Shen 1979). The hydrofoil is heuristically sized with the following equation.

$$S_{hf_{req}} = \frac{\rho_A V_s^2 S_w C_{L_{wmax}}}{\rho_W V_{hf_s}^2 C_{L_{hf_{max}}}} = 25 \left(\frac{\rho_A S_w C_{L_{wmax}}}{\rho_W C_{L_{hf_{max}}}}\right), \quad V_{hf_s} \coloneqq V_s/5$$

The rationale for this is that the hydrofoil should be able to lift the aircraft weight within 20% of the wing's stall speed as it accelerates, when the elevator is effective enough to provide moment corrections. This ensures the utility of the hydrofoil for a majority of the takeoff without increasing its size beyond the aircraft's dimensional constraints for its configuration on land. The hydrofoil configuration is a protrusion from the hull of the aircraft instead of a strut-based structure and to minimize design complexity. An anhedral angle of 20° has been selected and the hydrofoil, with its leading edge as the reference point, has been placed ahead of the CG by 50% of the MAC of the wing based on historical data, adjusting for the area accordingly:

$$S_{hf} = \frac{S_{hf_{\rm req}}}{\cos^2 \delta}$$

where  $\delta$  is the dihedral angle. The aspect ratio is selected as 5 for this analysis to reduce induced drag effects, which also satisfies dimensional constraints of the hydrofoil depth being lesser than the wheels of the landing gear. No sweep or taper has been included to simplify the analysis; sweep has benefits for subcavitating hydrofoils, but not supercavitating ones, and taper has negligible effects (Vagianos and Thurston 1952).

| Parameter              | Dimension   |  |
|------------------------|-------------|--|
| Hydrofoil Shape        | YS-920      |  |
| Area                   | $1.275 m^2$ |  |
| Aspect Ratio           | 5           |  |
| Span                   | 2.525 m     |  |
| Mean Aerodynamic Chord | 0.505 m     |  |
| Dihedral Angle         | -20°        |  |

|            |              | · •         |
|------------|--------------|-------------|
| Table 2: H | vdrofoil Spe | cifications |

The boundary layer mesh requirements for hydrodynamic flow are quite different from their aerodynamic counterpart. This estimation for  $y^+ < 1$  in both cases can be performed using the Schlichting skin-friction correlation, which is admissible for Reynolds numbers below  $10^9$ , providing the wall shear stress  $\tau_{wall}$ , shear velocity  $u^*$  and wall distance y as:

$$\tau_{M} = \frac{1}{2} \rho_{M} V_{\infty}^{2} [2 \log_{10}(Re_{x}) - 0.65]^{-2.3}$$
$$u_{M}^{*} = \sqrt{\frac{\tau_{M}}{\rho_{M}}}, \quad y_{M} = \frac{y^{+} \mu_{M}}{\rho_{M} u_{M}^{*}}$$

where *M* refers to the fluid medium. The ratio of the required wall distance for the hydrodynamic to the aerodynamic case to obtain  $y^+ \approx 1$ , with the constraint that both surfaces are travelling at the same speed, can be easily computed:

$$\frac{y_W}{y_A} = \frac{\mu_W/\mu_A}{\sqrt{(\rho_W/\rho_A)(\tau_W/\tau_A)}} = \frac{\mu_W/\mu_A}{\rho_W/\rho_A} \left[\frac{2\log_{10}(Re_W) - 0.65}{2\log_{10}(Re_A) - 0.65}\right]^{1.15} \approx 0.067$$

The first layer thickness requirement for the sized hydrofoil with the given computational limitations is  $y = 1 \times 10^{-4} m$ , so it is more effective to follow a decoupled approach with different meshes for the aerodynamic and hydrodynamic calculations. For a preliminary 2D analysis, a mesh size of approximately 60,000 elements will be analyzed.



A volume of fluid (VOF) approach is used to model the multi-phase cavitation flow with density and viscosity:

$$\rho = \gamma \rho_W + (1 - \gamma) \rho_V$$
  
$$\mu = \gamma \mu_W + (1 - \gamma) \mu_V$$

where  $\gamma = V_W/V$ , the ratio of the volume of water to the total control volume. *Cavitation Analysis* 

The coefficient of pressure  $(C_p)$  and Cavitation number  $(\sigma)$  are defined as:

$$C_p = \frac{P_{local} - P_{ref}}{\frac{1}{2}\rho_w V^2}, \quad \sigma = \frac{P_{ref} - P_{vap}}{\frac{1}{2}\rho_w V^2}$$

 $-C_{p_{min}} \ge \sigma$  is the defining condition for the development of cavitation over a surface travelling in water. Experiments and CFD simulations over hydrofoils have shown that the formation of cavitation results in an oscillatory fluctuation of the  $C_{L_{hf}}$  and  $C_{D_{hf}}$  values, with resulting variations in the  $(L/D)_{hf}$  ratio (Hong et al 2017). ANSYS Fluent provides the Schnerr-Sauer model by default for multiphase flow analyses for its numerical stability. *Takeoff Calculation Setup* 

The takeoff calculation is performed using a script developed in Python. The analysis is performed using a time-stepping approach. The geometry of the lifting surfaces is stored as a class and is utilized to calculate aerodynamic parameters such as the Reynolds number to minimize repeated operations, also providing ease of data transfer between different subprocesses. A thrust function of airspeed has been implemented using a quadratic interpolation method (Gudmundsson 2014):

$$T(V) = \left(\frac{T_{static} - 2T_{V_{max}}}{V_{max}^2}\right)V^2 + \left(\frac{3T_{V_{max}} - 2T_{static}}{V_{max}}\right)V + T_{static}$$



Fig 5: Thrust-Velocity Curve

## 2D Analysis

Before spending time and valuable computational resources on a complete, high-fidelity 3D analysis, a preliminary analysis based on 2D computations is advisable to account for possible errors and analysis of cavitation over the hydrofoil. It would also provide some insight into the procedure for more comprehensive analyses. The equation of motion to be solved for a 2D analysis in terms of non-dimensional coefficients is:

$$ma = T \cdot r - \frac{V^2}{2} \Big[ \rho_A S_w C_{d_w} + \rho_W S_{hf} C_{d_{hf}} \Big] - \theta(C_\Delta) \rho_W g B^3 C_R \frac{C_\Delta}{C_{\Delta_0}} - \theta(-C_\Delta) R_{Froude}$$
  
$$\frac{C_\Delta}{C_{\Delta_0}} = 1 - \frac{V^2}{2W} \Big[ \rho_A S_w C_{l_w} + \rho_W S_{hf} C_{l_{hf}} \Big]$$

where  $\theta(x)$  is the Heaviside step function and *r* is a factor representing pilot variation of the throttle as a function of time with the following definition:

$$r(t) = \begin{cases} 0.25 + 0.75(t/10), & 0 \le t < 10\\ 1, & t \ge 10 \end{cases}$$

Using Archimedes' principle, the buoyant force is defined as:

$$F_{buoy} = \rho_w g V_{hull}$$

Note that in the takeoff condition  $\Delta = F_{buoy}$ , which varies with speed during takeoff, justifying the correction term in the hull resistance.

The external forces in the vertical direction are balanced with the aircraft weight during takeoff until rotation, because the lifting forces increasing with airspeed reduce the contact volume of the aircraft with the water, thus reducing the buoyant force.

$$L_w + L_h + L_{hf} - F_{buoy} = mg$$

The hydrodynamic computations are performed in ANSYS Fluent to model cavitation effects, accounting for loss of lift and increase in drag more accurately than quicker solvers

such as XFOIL that do not solve Navier-Stokes equations and do not have cavitation models.

As the drag computation for the hull is effectively a 3D drag, suitable 3D approximations should be made to the 2D aerodynamic coefficients in the following manner:

1. The lift coefficients for the airfoil ( $C_l$ ) are scaled down to 90% per iteration for the lift coefficients of the wing ( $C_L$ ) to account for induced drag effects.

2. The drag coefficients for the wing  $(C_D)$  are computed using the profile drag  $(C_{d_0})$ , drag polar  $(C_{d_n})$  and induced drag  $(C_{D_i})$ .

$$C_D = C_{d_0} + k_1 (C_l - C_{l_{\alpha=0}})^2 + K (C_l - C_{l_{\alpha=0}})^2, \quad K = \frac{1}{\pi e A R}$$

The termination condition for the calculation is the aircraft attaining the takeoff speed, defined as 1.2 times the stall speed:

$$V_{TO} := 1.2V_s = 1.2 \sqrt{\frac{2W}{\rho_A S_w C_{L_{max}}}}$$

 $C_{L_{max}}$  is chosen from an airfoil analysis performed in XFOIL during the preliminary design of the aircraft, modified appropriately for a 3D wing. The freestream velocity to be fed into the aerodynamic and hydrodynamic solvers is calculated by numerical integration. The position is calculated using the average velocity from the previous and current step.

$$v_i = v_{i-1} + a_i(t_i - t_{i-1})$$
  
$$x_i = x_{i-1} + \frac{1}{2}(u_i + u_{i-1})(t_i - t_{i-1})$$

The setup updates the velocity every second, so the CFD solvers take the input velocity in each iteration and run the simulation time for 1 second, providing the aerodynamic/hydrodynamic forces which are then substituted into the equations of motion. The longitudinal moments are calculated using the following equation.

$$M_{y}^{b} = M_{ac_{w}} - Ty_{t} - L_{h}\cos\alpha_{h} |x_{ac_{h}} - x_{cg}| + [D_{w}\cos\alpha - L_{w}\sin\alpha]y_{w}$$
  
+  $[L_{w}\cos\alpha + D_{w}\sin\alpha]|x_{cg} - x_{ac_{w}}| + [L_{hf}\cos\alpha_{hf} + D_{hf}\sin\alpha]|x_{cg} - x_{ac_{hf}}|$   
-  $[D_{hf}\cos\alpha_{hf} - L_{hf}\sin\alpha_{hf}]y_{hf}$ 

where the geometric parameters are obtained from the aircraft design and sizing process, and the aerodynamic and hydrodynamic forces are obtained from the CFD solvers.  $y_{ac_{hf}}$ and  $x_{ac_{hf}}$  represent the vertical and horizontal positions of the aerodynamic center of the hydrofoil; these are variables in the takeoff analysis for an optimization scheme, which will be elaborated upon in a later section. The expected result is the aircraft with hydrofoil configuration taking off in lesser distance and time than without one.

# 4. RESULTS

#### **Cruise Conditions**

Lifting surface characteristics at cruise conditions have computed using an automated script that calls XFOIL with a Clark-Y hydrofoil, selected based on available literature from (Hong et al 2017), due to XFOIL's computational limitations:



Fig 6: Aerodynamic Characteristics at Cruise

## 2D Analysis

The following calculations were performed for 2D meshes using a completely automated setup developed as a Python script, which generates ANSYS Fluent journals based on the velocity and angle of attack.

The following simulation uses ANSYS Fluent for the aircraft with no hydrofoil. Each loop runs for 1500 iterations under steady-state conditions due to computational limitations.

These results indicate that the aircraft reaches its takeoff velocity of 44 m/s in approximately 33 seconds with its load coefficient reduced to approximately 1/6 of its original value, which is reasonable to some extent given the assumptions. A longer time is expected than the result because of inaccurate hull drag modelling.



Fig 7: No-Hydrofoil Configuration - Fluent Aerodynamic Results



Fig 8: No-Hydrofoil Configuration: Fluent Takeoff Results



#### Fig 9: No-Hydrofoil Configuration: Fluent Takeoff Analysis

The following shows the results of Fluent simulations with the hydrofoil configuration. The setup for the hydrofoil is a transient simulation with timestep  $10^{-4}$  to avoid divergences. The maximum number of iterations per timestep is 50 for convergence, and the total number of timesteps is 250 for a total simulation time of 0.025 seconds per loop due to computational restrictions.



#### Fig 10: Hydrofoil Configuration: Fluent Aerodynamic Results

These results predict that the drag generated by the hydrofoil eventually causes the airplane to reach a terminal velocity of approximately 16.4 m/s. The hydrofoil also produces the major lifting forces to reduce the buoyant forces to 0, when the aircraft is effectively hydroplaning on the water surface.







Fig 12: Hydrofoil Configuration: Takeoff Analysis

# 5. CONCLUSIONS

A preliminary 2D analysis with 3D corrections using CFD simulations shows that a design that incorporates a fully submerged hydrofoil for the entire duration of the takeoff run does not reduce the time required to achieve takeoff velocity or the takeoff distance, rather it

does the opposite. This is due to the large drag generated by the hydrofoil due to cavitation effects:



Fig 13: Cavitation over Hydrofoil at Terminal Velocity

The modelling of ventilation is not included in this paper, which loses some of the important physics that takes place in the real-world process. Air-water interface modelling will provide the necessary setup for simulating ventilation effects.

#### Future Work

A drawback of the hydrodynamic setup is that interference effects between the hydrofoil and the hull are neglected, which are bound to play an important role in hydrodynamic drag calculations. A setup which calculates hull resistance coupled with a hydrofoil including cavitation studies would provide rich literature.

#### **Optimization Frameworks**

As mentioned before, hulls are designed to keep the trimming moment close to 0 at low speeds. The inclusion of a hydrofoil at some distance from the CG induces moments about the aircraft. As the lift generated by hydrofoils in water is greater than lifting surfaces in air at lower speeds, an optimization setup is required to determine the placement of the hydrofoil to minimize moments at low speeds while maximizing lift-to-drag ratio.

| Optimization        | Function variables         | Description                         |
|---------------------|----------------------------|-------------------------------------|
| Maximize            | $(L/D)_{hf}$               | Lift-to-drag ratio of the hydrofoil |
| Design<br>variables | $\alpha_{hf}$              | Angle of attack of the hydrofoil    |
|                     | $V_{\infty}$               | Freestream velocity                 |
|                     | $(x_{ac}, y_{ac})$         | Hydrofoil coordinates               |
|                     | $0 < x_{ac} < x_{cg}$      | Horizontal position                 |
| Constraints         | $y_{1a} < y_{ac} < y_{ca}$ | Vertical position                   |
|                     |                            | Longitudinal moment                 |

#### Table 3: Optimization Framework

#### The 2018 Structures Congress (Structures18)

Songdo Convensia, Incheon, Korea, August 27 - 31, 2018

$$M \approx 0$$

## 3D Cavitation Analysis

A preliminary 3D cavitation analysis has been performed over a Clark-Y hydrofoil in preparation to set up the takeoff calculator for 3D analyses. The following results evidently display the cavitation phenomenon taking place on the hydrofoil.



Fig 14: Cavitation over the Clark-Y 3D Hydrofoil

The automated setup will require application of Fast Fourier Transforms that measure the  $C_L$  and  $C_D$  variations over a period, which will then be averaged over.

## 6. ACKNOWLEDGEMENTS

The presenting author would like to thank Prof. Rhea P. Liem for her guidance and support through the development of this research.

## 7. **REFERENCES**

- Qiu, L., and Song, W. (2012), "Efficient decoupled hydrodynamic and aerodynamic analysis of amphibian aircraft water take-off process.", *Journal of Aircraft*, **50** (5), 1369-1379.
- Eppler, R. and Shen, Y. T. (1979), "Wing Sections for Hydrofoils Part 1: Symmetrical Profiles", *Journal of Ship Research*, **23** (3), 209-217.
- Latorre, R. and Teerasin, S. (1992), "Calculation of Hydrofoil Craft Take-off Speed Including Influence of Foil Size, Foil Angle and Propeller Shaft Angle", *Ocean Engineering*, **19** (2), 183-197.
- Larsson, L. and Raven, H. C. (2010), *Ship Resistance and Flow.*, Paulling, R. J. Ed., Principles of Naval Architecture Series, Jersey City: Society of Naval Architects and Marine Engineers, New Jersey, USA.
- Garg, N., Kenway, G.K., Lyu, Z., Martins, J.R. and Young, Y.L. (2015a), "High-fidelity hydrodynamic shape optimization of a 3-D hydrofoil", *Journal of Ship Research*, **59** (4), 209–226.

- Garg, N., Kenway, G.K., Lyu, Z., Martins, J.R. and Young, Y.L. (2017), "High-fidelity multipoint hydrostructural optimization of a 3-D hydrofoil", *Journal of Fluids and Structures*, **71**, 15-39.
- Gudmundsson, S. (2014), *General Aviation Aircraft Design: Applied Methods and Procedures*, Elsevier Inc., Waltham, MA, USA.
- Suydam, H.B. (1952), Hydrodynamic Characteristics of a Low-Drag, Planing-Tail Flying-Boat Hull, NACA Technical Note 2481, Institution, City, Country.
- Hugli Jr., W.C. and Axt, W. C. (1951), Hydrodynamic Investigation of a Series of Hull Models Suitable for Small Flying Boats and Amphibians, NACA Technical Note 2503., Institution, City, Country.
- Frisk, D. and Tegehall, L. (2015), "Prediction of High-Speed Planing Hull Resistance and Running Attitude", M.Sc. Dissertation, Chalmers University of Technology, Gothenburg, Sweden.
- Vagianos, N.J. and Thurston, D. B. (1970), Hydrofoil Seaplane Design, Report No. 6912, Thurston Aircraft Corporation, Maine, USA.
- Hong, F., Yuan J. and Zhou, B. (2017), "Application of a new cavitation model for computations of unsteady turbulent cavitating flows around a hydrofoil", *Journal of Mechanical Science and Technology*, **31** (1), 249-260.
- Roohi, E., Zahiri A. P. and Passandideh-Fard, M. (2013), "Numerical simulation of cavitation around a two-dimensional hydrofoil using VOF method and LES turbulence model", *Applied Mathematical Modelling*, **37** (9), 6469-6488.