The Hydrodynamics of the WIG (Wing-In-Ground) Effect Craft

†B.C. Khoo[1], and H.B. Koe[1]

[1]Department of Mechanical Engineering, National University of Singapore, Singapore.

†Presenting and corresponding author: mpekbc@nus.edu.sg

Abstract

Due to its close proximity to the surface of water, Wing-In-Ground (WIG) crafts have much benefits from the increased aerodynamic efficiency which easily translates into fuel saving. This is besides the much higher speed possible of up to O(100) nautical miles per hour for the WIG craft which is at least twice the speed of the fastest water-surface speed boat of less than 50 nautical miles per hour. However, a WIG craft needs to be able to overcome significant hydrodynamic drag to take-off from water. The objective of this work is to investigate the hydrodynamics of a WIG craft through towed-tank test experiment as well as computational fluid dynamics simulation. From the model test, the resistance, sinkage, running trim angle and wetted area are obtained throughout the take-off speed range. Region associated with the highest resistance called hump drag is identified as well as the possibility of secondary hump and slight oscillation are discussed. Despite the complex FSI (Fluid-Structure Interaction) between the hull and water, the good comparison between experiment and simulation shows that the present state-of-the art numerical simulation is a powerful tool for WIG craft designers. An important finding is the critical presence of the stepped hull in overcoming the mentioned humped drag.

Keywords: Wing-in-Ground, Hydrodynamics, Stepped Hulls, Towing Test, CFD

Introduction

In the search of an efficient marine transportation, Wing-In-Ground (WIG) craft provides a promising solution. It is comfortable since it's flying above the sea, thus away from the wavy seas. It has a large Lift to Drag ratio, which means WIG craft has an efficient aerodynamics form due to its close proximity to the surface of the water where the ground effect takes place [1]. These means that WIG is an attractive vehicle for commercial application.

Apart from aerodynamics differences with aircraft due to ground effect [2], the hydrodynamics of the craft is another important aspect of WIG design since it needs to take-off from water. During take-off phase the hydrodynamic drag can be relatively very high compared to the aerodynamics drag, thus this often leads to high take-off thrust requirement. Initially, most of the weight is supported by water through buoyancy, as the craft speed ups the hydrodynamics drag builds up and reaches a maximum point which is called Hump Drag. It is generally occurs between 30 - 50% of the take-off speed [3]. At faster speed, the aerodynamics lift becomes significant enough to lift the craft and the hydrodynamic drag starts to reduce gradually up to the take-off point.

In order to make sure the hydrodynamics drag at high speed is acceptable, a stepped hull design is employed. The shape of the WIG craft hull is similar to those of high speed planing boat with a sudden discontinuity called step located amidships. Flying boat has been employing such design and interested reader can refer to [4]. The sudden discontinuity might induces high hydrodynamics drag, however, at high speed the flow will be separated from it and thus reducing the wetted area significantly which in turn makes the total drag is acceptable. If the step is nonexistent, as shown in [5], a planing hull shape will tend to have small trim angle at high speed which translated to significant viscous drag arising from the wetted area. As compared to planing hull where semi-empirical method for analysis is available [6], stepped hull flow is more complex and this makes performance prediction more challenging and typically done through towing test which is costly and time consuming.

The objective of this work is to study the hydrodynamics of a WIG craft by employing both experimental method which is done via towing test and numerical method through computational fluid dynamics (CFD). The experimental method also serves as a benchmark in validating the solution that is obtained from numerical simulation, which will help to check the viability of CFD as a design tool which will reduce significantly the design cost. The value of drag, trim, sinkage and the wetted area during take-off are presented. Furthermore, a detail discussion is provided on the important hydrodynamics features such as the hump drag, possibility of secondary hump as well as porpoising instability.

Experimental Method

Model Description

A scaled model of AirFish-8 WIG craft is built and tested in the Davidson Laboratory towing tank at the Stevens Institute of Technology as shown in Fig. 1. The geometry of the model is given in Fig. 2 and the principal dimension in Table 1. For the study, only the parts which is important to the hydrodynamics of the craft are built, notably the main fuselage hull and the two floats called sponsons. The hull and sponsons are of a hard chine type which is primarily used for high speed boat and there is a step amidships of the main hull. The step divides the hull into two regions, forebody which is the part in front and afterbody which is the part behind the step. The model is made primarily from fiberglass and an aluminum structure is built to make sure there is a rigid connection between the hull and sponsons which is 0.8 m away from the symmetry plane of the hull.

Figure 1: Model Tested

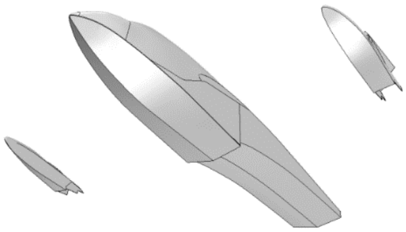

Figure 2: 3D Cad of the WIG Hull

Table 1: Model Principal Properties

Parameter	Symbol	Value
Overall Length	LOA	1.93 m
Beam	b	0.29 m
Deadrise	β	12 degree
Displacement	Δ_0	13.12 kg
Longitudinal Center of Gravity	LCG	0.90 m

Testing Procedure

The towing test is done similar to flying boat towing test as discussed in [7]. Here, the aerodynamics forces contribution is modelled in the tank. The lift is provided through the parabolic unloading method following [7] which is essentially the same as assuming a constant lift coefficient (C_L). Aerodynamic moment coefficient (C_m) is assumed constant. Pitch damping from the tail is calculated as in [8] and is given by using a damper filled with oil and calibrated accordingly. The towing pivot is located at the LCG location and 0.26 m above the keel to simulate the propeller point of action. The models were free to trim and heave, but is restricted in the yaw, roll, surge and sway. The sinkage was measured using a motion transducer attached, trim was measured using an inclinometer mounted on the model and the drag was measured using a drag balance. Free to trim and free to heave test were carried out in the speed coefficient (C_V) ranging from 0.58 to 5.21 which covers most of the take-off speed range. The speed coefficient is defined as

$$C_V = \frac{V}{\sqrt{gb}}. \tag{1}$$

Here V is the speed and g is the gravitational acceleration. Fixed trim test were also carried out at near take-off speed. The time histories of drag, sinkage and trim are captured as well as underwater photos to determine the wetted area. Here, the hydrodynamics drag is the interest, the aerodynamics of the model is captured by running the model above the water and then deducted from the total drag as in [9].

Numerical Method

Governing Equations

The governing equations solved by the software are described by the Reynold-averaged Navier-Stokes (RANS) equations which in incompressible fluid is given below:

$$\frac{\partial}{\partial x_i}(\bar{u}_i) = 0 \tag{2}$$

$$\frac{\partial}{\partial t}(\rho\bar{u}_i) + \frac{\partial}{\partial x_j}\left(\rho\overline{u_i u_j} + \bar{p}\delta_{ij} - \mu\frac{\partial \bar{u}_i}{\partial x_j} + \rho\overline{u_i' u_j'}\right) = S_i. \tag{3}$$

Here, ρ is the density of the fluid, u_i is the fluid velocity component, p is the pressure, and S is the source term. The SST (Menter) K-ω is chosen as the turbulence model. In the simulation, both air and water are simultaneously simulated using the Volume of Fraction (VOF) approach whereby an additional convection equation is solved in the domain where the scalar solved is the volume fraction. That is,

$$\rho = \alpha\rho_{water} + (1-\alpha)\rho_{air} \tag{4}$$

$$\mu = \alpha\mu_{water} + (1-\alpha)\mu_{air} \tag{5}$$

$$\frac{\partial \alpha}{\partial t} + \frac{\partial}{\partial x_i}(\alpha u_i) = S_\alpha \tag{6}$$

where α is the volume fraction of water and $\alpha = 0.5$ is used when visualizing the free surface. HRIC (High Resolution Interface Capturing) Method is used in discretizing the VOF flux. Implicit unsteady and segregated flow technique is then employed to solve these equations. The details on how the software is employed to solve these equations are given in [10].

Mesh and Computational Domain

The domain as shown in Fig. 3 is only half since flow symmetry is assumed. The reference of $x = 0$ is located at the bow while positive x-axis means downstream, $z = 0$ is assumed to be the calm water surface position while positive z-axis means positive vertical. The size of the domain is given by $-4\ m \leq x \leq 10\ m$, $0 \leq y \leq 2.45\ m$ and $-2\ m \leq z \leq 1.5\ m$. In order to tackle the issue of large hull movement (in heave and pitch), the overset mesh methodology is utilized. In this method, the domain mesh is fixed while an additional domain to encapsulate the model will move together with the model. Additional refinement is used near z = 0 to capture the free surface accurately. Refinement of mesh in the wake region on both the main hull and sponson as well as step area is used. The mesh close to the wall is designed to make sure the value of $y+\leq 80$ is adhered to. The typical mesh used is given in Fig.4.

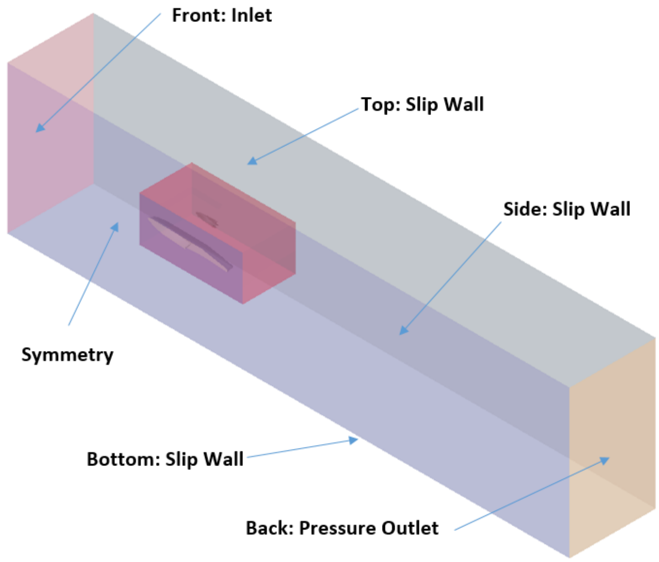

Figure 3: Simulation Domain

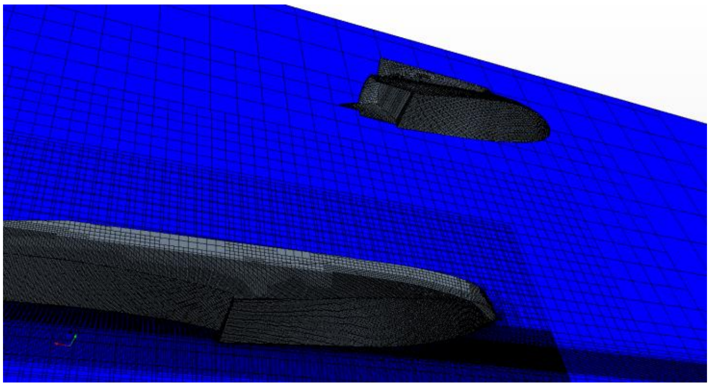

Figure 4: Mesh refinement around the hull.

In order to make sure that the solution has converged well, a mesh sensitivity study is carried to make sure all the relevant complex flow phenomena are captured. Table 2 presented the study with three different mesh configurations. The difference in the computed value of D/W (non-dimensional drag), trim angle and sinkage between them are less than 5%. The medium mesh is then chosen so that the computational cost is still acceptable.

Table 2: Mesh independence study

Mesh	No. of Cells	D/W	Trim (deg)	Sinkage (beam)
Coarse	1.1 Millions	0.172	4.40	0.081
Medium	1.8 Millions	0.171	4.44	0.081
Fine	3.6 Millions	0.175	4.54	0.082

Result and Discussion

Fig. 5 shows the hydrodynamic drag of the WIG craft that is obtained from through the free-to-trim test. The attitude of the craft in water described by the trim and sinkage are shown in Fig. 6 and Fig. 7, respectively. Here, the drag (D) is non-dimensionalized with respect to the weight (W) of the craft. It is observed that the characteristics of the drag are different from a typical high speed boat where it typically increases as the speed increased. In WIG craft the drag increases, but will reach a maximum drag called hump drag at $C_V = 1.7$. Once this speed is passed, the drag starts to reduce towards the take-off speed at $C_V = 5.8$. Here, one observes that the CFD solution is able to produce a good agreement on drag with the tow tank result both qualitatively in term of trend and quantitatively. The absolute error between CFD and tow test result has an average of 8.4%. CFD solution is able to predict the existence of hump drag at $C_V = 1.7$ despite slightly over predicting the drag value. The reduction of drag is primarily caused by the dominance of aerodynamics lift at higher speed which reduces the load on water as well as the existence of the step.

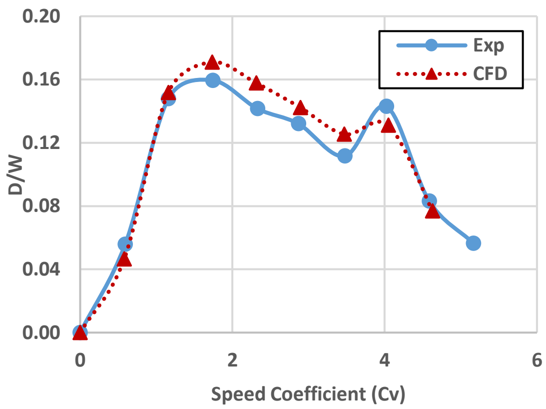

Figure 5: Drag vs. Speed Comparison.

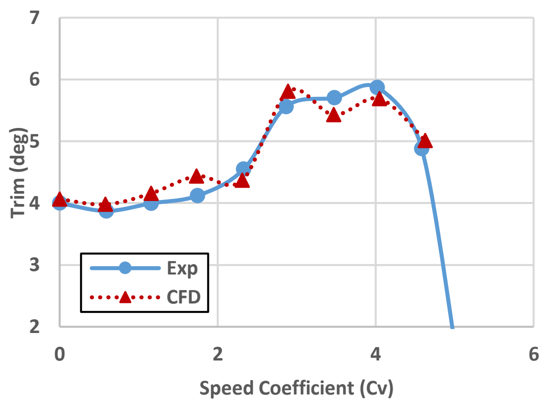

Figure 6: Trim vs. Speed Comparison.

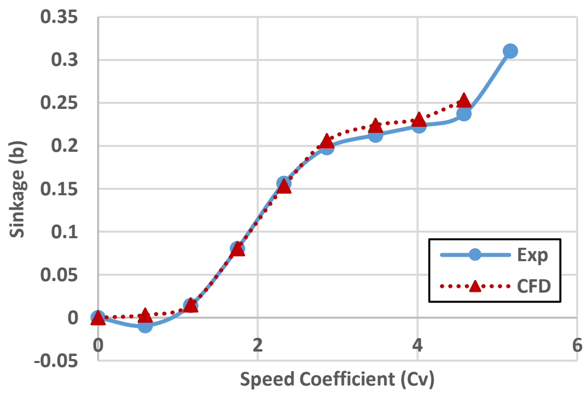

Figure 7: Sinkage vs. Speed Comparison.

Interestingly, there exists another local maximum on the drag curve at $C_V = 4$, this is called the secondary hump. This is in fact not unique to WIG craft, as can be seen from towing test done on a series flying boat done by NACA [11]. If this secondary hump is higher than the available thrust, it will prevent the craft from taking-off. One reason provided in [12] is that as the speed is getting faster, the wetted beam at the step is getting smaller than the actual beam and as a result a heavy spray escapes backward (typically called blister spray). The tangential contact of this spray with the hull (afterbody) will increase the frictional resistance significantly. CFD also shows the secondary hump despite slightly under predicting it. It was found that a proper mesh resolution is needed in order to capture the blister spray emanating from the step since the spray dimension is thin. Table 3 shows three different mesh configuration where for meshes 2 and 3 additional refinements added in the area behind the step to capture the blister spray. On Mesh 1 with the lowest resolution, there is under prediction of D/W of 22.9% and on the highest mesh resolution the error reduced to only 8.5%. Fig. 8 reveals that as the mesh is refined, the simulation seems to capture the area wetted by the blister spray better. Hence it is important for WIG craft designers to make sure that the simulation capture the blister spray correctly to make sure whether the secondary hump exist on certain design or not. Based on Mesh 3, mesh refinement of 0.5% beam around the blister spray is recommended.

Table 3: Secondary hump mesh sensitivity study.

Mesh	No. of Cells	D/W	Error (%)
1	2 Millions	0.110	22.9
2	3.5 Millions	0.127	11.4
3	11 Millions	0.131	8.5
Towing Test		0.143	

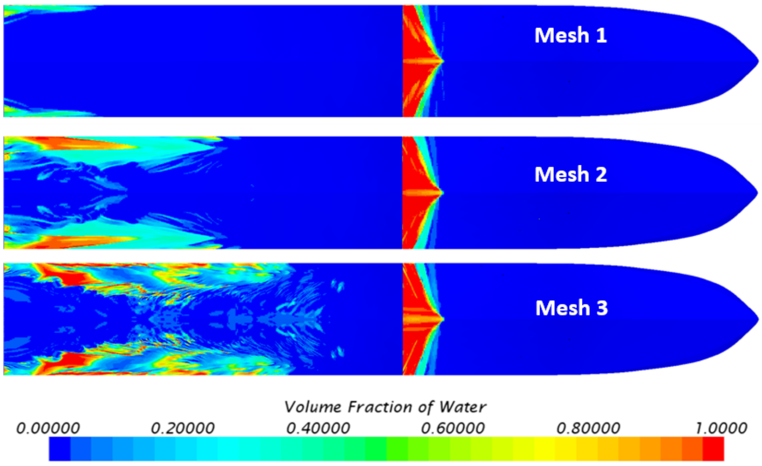

Figure 8: Volume fraction of Water on the fuselage at Cv = 4 on different mesh.

In investigating the secondary hump, the result of fix trim test done at this particular speed ($C_V = 4$) reveals the general behavior of the WIG craft hull. Fig. 9 shows the hydrodynamic drag behavior at different trim angle. The lowest drag of $D/W = 0.118$ is found when $\tau = 4.1°$ while for the free-to-trim test result in $\tau = 5.8°$ and $D/W = 0.143$, a 21% increase in drag. This means that if the WIG is to maintain the trim at the optimum trim angle, the secondary hump will not be seen. Slight oscillation is also found in both towing tank test and simulation near the secondary hump speed. Several researchers Savitsky & Morabito [5]; Garland [13] suggest that the blister spray is able to create such instability on planing hull. However, the amplitude of the oscillation of 0.5° is still acceptable [14]. Moreover, during the take-off process this phase will pass quickly since the craft is accelerating and thus deemed to be acceptable.

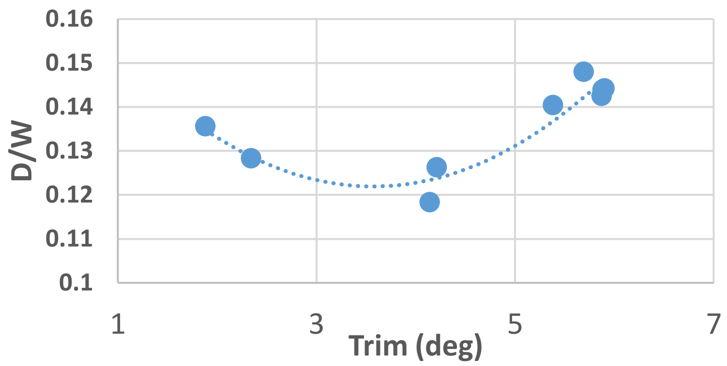

Figure 9: Drag on varying trim at Cv = 4.

The comparison on the equilibrium trim (τ) condition is given in Fig. 6 The tangent to the forebody keel at the step is used as a reference for the trim angle. Generally, there is a good agreement between the tow tank and CFD solution with an average error of only 4.3% which is 0.2° in absolute number. WIG craft trim does not vary significantly during the take-off process, unlike typical high speed boat where the trim will get smaller as the speed gets higher [5]. The trim is slowly increasing from 4° to the maximum of 6° at $C_V = 3.5$. The reduction in trim seen when $C_V \geq 4$ which is near the take-off speed is primarily due to the pitching down moment that the aerodynamics surface provided. At this speed the WIG craft can be trimmed up for taking-off. The comparison of sinkage which measures the center of gravity movement (in vertical axis) w.r.t the static condition is given in Fig. 7. Here, it is non-dimensionalized with respect to the beam (b) of the main hull. During the displacement mode (stationary) where the hull is mainly supported by buoyancy, the sinkage is small. Once the dynamic lift of the planing hull build up there is an appreciable increase in sinkage as seen when $C_V > 1$. There is also a good comparison between the numerical and experimental data with average error of 3.8%. The comparisons on trim and sinkage show that CFD is able to produce the equilibrium state accurately and is capable of simulating the complex fluid structure interaction between the fluid and the WIG hull.

The wetted area at $C_V = 1.7$ and $C_V = 4$ are given by Fig. 10 and Fig. 11, respectively. In each figure, the top half is the underwater photo taken in the tank while the bottom half is the underwater perspective obtained from CFD. At $C_V = 1.7$ which corresponds to hump speed, the step is already aerated and the wake from the forebody hit the afterbody again. As the craft speeds up, there is less weight on the water and higher pressure, thus less wetted area is needed to support the craft. In both comparisons, CFD is able to reproduce the wetted area, this means that the numerical method is able to predict the flow separation at the step, the wake generated by the forebody and the re-attachment point at the afterbody correctly. From the comparison on drag, trim, sinkage and wetted area with the towing tank, it is concluded that the present state of simulation technology is able to reproduce quite accurately the physics of WIG craft hydrodynamics. This means that CFD is a very powerful tool that designer can use when designing WIG craft before embarking on the experimental towing test which is costly and time consuming.

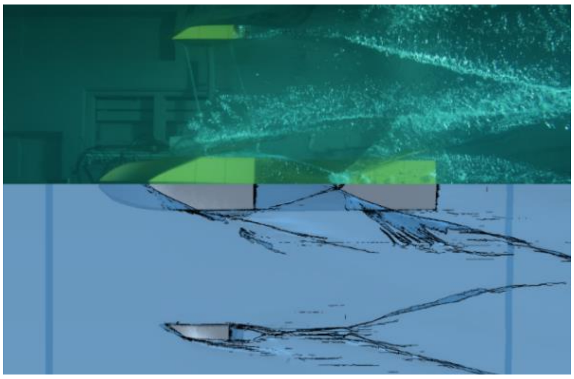

Figure 10: Wetted area comparison at Cv = 1.7. Top is tow test and bottom is CFD.

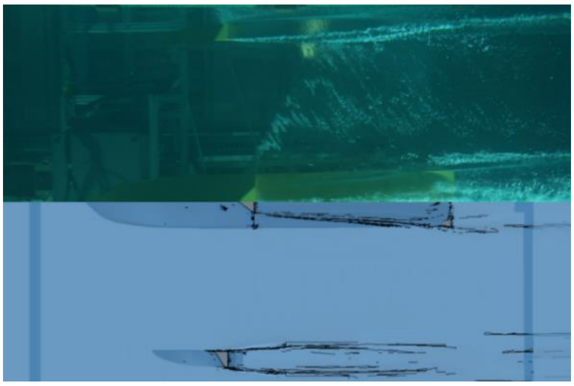

Figure 11: Wetted area comparison at Cv = 4.0. Top is tow test and bottom is CFD.

In order to gain some understanding on the use of step on WIG craft. A comparison against (hypothetical) hull without step and no sponsons as well as purely prismatic is performed here. It is assumed that this hull has the same beam (b), deadrise (β), displacement (Δ_0), LCG and the same parabolic unloading for simulating aerodynamics lift. Semi-empirical procedure based on [6] with the lift and moment coefficient obtained from [15] is used. The non-dimensional drag (D/W) is then calculated and given in Fig. 12. Without step, the drag at lower speed is lower ($C_V = 2$) but then it rises dramatically by 190% higher at $C_V = 4$. The main reason of this is the trim angle turns out to be very low ($\leq 1.5°$) and this makes the surface of the hull to be wetted and resulting in high viscous drag. On the other hand, the trim of the stepped WIG craft hull does not vary significantly as shown in Fig. 6 which results in a more optimum trim angle. Hence, despite the hull without step has lower drag at low speed, it is impractical to be used for WIG craft since high drag at high speed means that a very powerful engine is needed to take-off.

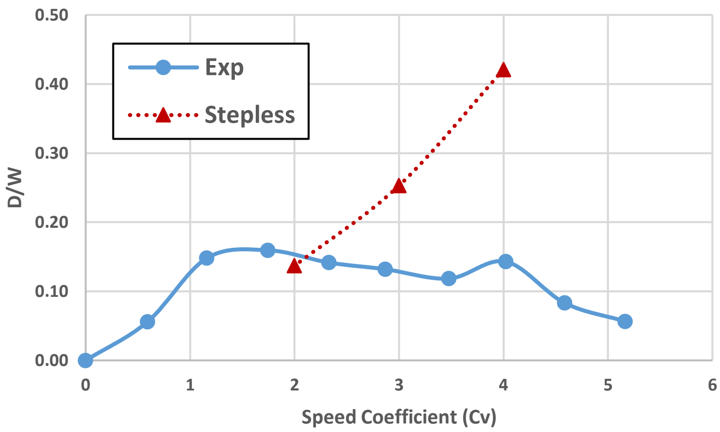

Figure 12: Comparison of Airfish-8 against a stepless hull.

Conclusion

Detailed investigation on the hydrodynamics of Wing-In-Ground craft has been performed by using towing test experiment and CFD. Through the validation of the numerical method, the current state-of-the-art CFD is a promising design tool for WIG craft designers. Several important features such as hump drag, secondary hump as well as slight oscillation are identified both experimentally and numerically. The blister spray emanating from the step is found to be the possible source of increase in drag at secondary hump. It has been shown as well that this secondary hump can be avoided when the hull is running at its optimum trim angle. Slight oscillation was found as well near the region where secondary hump appears, albeit it is considered acceptable, the study is important during the design process in order to make sure that the WIG craft is able to take-off.

References

[1] K. V. Rozhdestvensky, "Wing-In-Ground Effect Vehicles," *Progress of Aerospace Science,* vol. 42, pp. 211-283, 2006.

[2] R. Irodov, "Criteria of the Longituinal Stability of the Ekranoplane," *Ucheniye Zapiski Tsagi,* vol. 1, no. 4, pp. 63-74, 1970.

[3] D. Ingo and L. M. de, "Hydrodynamic Characteristics of Seaplanes as Affected by Hull Shape Parameters," *AIAA Intersociety Advanced Marine Vehicle Conference,* p. 275, 1989.

[4] E. Stout, "Development of High-Speed Water-Based Aircraft," *Journal of Aeronautical Science,* vol. 17, no. 8, pp. 457-480, 1950.

[5] D. Savitsky and M. Morabito, "Surface Wave Contours Associated With The Forebody Wake of Stepped Planing Hulls," *Marine Technology,* vol. 47, no. 1, pp. 1-16, 2010.

[6] D. Savitsky, "Hydrodynamic Design of Planing hulls," *Marine Technology,* vol. 1, no. 1, pp. 71-95, 1964.

[7] H. J. Wc and W. Axt, "Hydrodynamic investigation of a series of hull models suitable for small flying boats and amphibians," Stevens Institute of Technology, New Jersey, 1951.

[8] L. Chu, "Damping Coefficients Due To Tail Surfaces," *NACA TR 136,* 1923.

[9] W. H. Jr and W. C. Axt, "Hydrodynamic Investigation of a Series of Hull Models Suitable for Small Flying Boats and Amphibians," *NACA TN 2503,* 1961.

[10] CD-Adapco, "Star-CCM+ v10.04 user Guide," 2015.

[11] J. Allison, "The Effect of The Angle of Afterbody Keel On The Water Performance of a Flying-Boat Hull Model," *NACA TN 541,* 1935.

[12] W. Sottorf, "The Design of Floats," *NACA TM 860,* 1938.

[13] R. W. Garland, "Stepped Planing Hull Investigation," *SNAME Transaction,* 2011.

[14] A. Smith and H. White, "A Review of Porpoising Instability of Seaplanes," *Royal Aircraft Establishment,* vol. 2, no. 2, 1954.

[15] P. W. Brown, "An Experimental and Theoretical Study of Planing Surfaces with Trim Flaps," SIT Report SIT-DL-71-1463, Hoboken, New Jersey, 1971.