33RD CONGRESS
OF THE INTERNATIONAL COUNCIL
OF THE AERONAUTICAL SCIENCES
STOCKHOLM, SWEDEN, 4-9 SEPTEMBER, 2022

ICAS 2022 SWEDEN

CFD TAKE-OFF ANALYSIS OF A SEAPLANE HULL THROUGH A DYNAMIC MULTIPHASE MODEL

Sinisi M.[1], Cantanna G.[1], Occhipinti G.[1], Caramazza A.[1], Dilonardo G.[1] & Cestino E.[1]

[1]Politecnico di Torino, Department of Aerospace Engineering, Turin, Italy Corso Duca degli Abruzzi 24, 10129, Turin, Italy

Abstract

Nowadays, marine economy shows a renewed interest in seaplanes thanks to their multiplex usage. Hull geometries remains almost the same since the third decade of the last century, when models such S55-X appeared. The main progress has consisted in a step behind the forebody on the lower surface of the hull, to decrease roughly the hydrodynamic drag in order to make a rapid take-off. Thus, the analysis provides the fluid dynamic behaviour in the take-off flight configuration of one of the twin hulls of the seaplane Savoia-Marchetti S55-X model aircraft built by the student team "Team S55". The study aims showing the analytical method useful to evaluate the velocity field, the pressure field, the trust and the drag coefficient changes on a hull initially at rest reaching the take-off velocity. The analytical results are compared with experimental data collected by NASA through the S55-X tank registered in the Technical Note no.635.

Keywords: Seaplane, CFD take-off analysis, Overset mesh, VOF Wave, DFBI

1. Introduction

The Team S55 of the Politecnico di Torino is a student team, managed by Professor Cestino Enrico from Aerospace and Mechanic Engineering Department. The Team's goal is designing, making and testing a flying replica of the historical seaplane Savoia-Marchetti S55X, scaled 1:8. The Aerodynamics Section of the team has been focused on the study of the aerodynamic behaviour of the seaplane, thanks to the use of Computational Fluid Dynamics (CFD) software. The paper describes the methodologies applied for a CFD take-off analysis and its results.

S-55-X has been an Italian pride: it is well known for its transoceanic flight in 1933, scoring a great technical advance in flight history. Its design was unusual, having a concave bottom of the forebody of the hull. The peculiar shape of the hull minimizes the dead rise decreasing a lot the drag, almost like a modern foiled sailboat [9][12]. As demonstrated in N.A.C.A.'s T/N 635 [2], a flat gliding surface reduces the height of the wake profile, and a concave bottom reduces the height of the transverse bow wave. They adopted the model with fixed trim and heave to examine the effects of velocity, trim, and draft on hydrodynamic performance. The increasing of the resistance at high-speed range is small. Thus, S-55-X shows a very clean running at all speeds differently than its competitors.

The design of a seaplane is a compromise between the desired hydrodynamic performance and the aerodynamic efficiency, often causing the first one a loss in the second and making difficult to estimate the coupled effects. Besides the fuselage geometry that disturbs water into wave flows, the hydrodynamic resistance mainly depends on speed, water load, and trim angle, which is defined as the angle between the keel of forebody and the water level [1]. The water load and trim angle would be influenced by the aerodynamic lift and pitching moment respectively. However, the numerical investigations on the take-off performance of seaplanes combining the analysis of aerodynamics and hydrodynamics have been rarely published.

The paper aims to investigate numerically the drag performance of a simplified 3-D model during the take-off. Indeed, the lift has to exceed the weight to rise the air, and the total drag has to be won by the thrust. The condition is even stricter in case of hydroplanes because air drag and water drag add

up. The drag is expected to increase while the hydroplane starts moving, until the lift grows up and alleviates the effect of gravity.
The most interesting phenomena during the take-off take place at the interface between the fluid and the hull, and at the wake. According to [2], N.A.C.A. has stated that the deadrise of the afterbody increases with distance after of the step, thus a wave rises at the step and evolves in the wake. The take-off simulation is complex especially in case of more than one fluid involved, because it requires several analytical models to grasp the actual interactions made by the fluids, and it is very expensive from the computational point of view. Thus, the analysis considers only a single hull of the aircraft.
The analysis consists of three work-phase: pre-processing, processing and post-processing. The pre-processing involves the geometry preparation and clean up and the mesh creation, while the processing interests the setting of both physical and computational models, and the post-processing is the request for reports and plots to read results.

2. Take-off CFD analysis

Take-off is the most critical phase of flight, during the ascend. Since this phase is a constraint on an aircraft during the initial design process, preliminary CFD computations are essential; they are provided by dedicated softwares. Because of the changes occurring in such a phase, modeling the take-off throughout a software is quite complex.
The analysis has been structured in two different parts.
The first one is focused on the study of the last moments of the take-off, in particular the phase in which the hull starts rising from the water surface. The main challenge of this work is to simulate the translational movement of the hull and, at the same time, to model the multi-phase fluid around the body.
The second part of the analysis aims to simulate the experimental work made by the N.A.C.A.'s study on a 1:5.25 replica of S-55-X hull in 1951 [2] and to compare it to the CFD study on a 1:8 replica made by Team S55.
Building the analysis, the first step is modelling forces, since the hull has not lifting surfaces itself. In this case lift and weight behave like external loads.
The take-off speed considered for this simulation is equal to 18 m/s.
According to the goal of comparing analytic and experimental data, the simulated conditions of the experiments are obtained assuming the same N.A.C.A.'s water load coefficient. The load coefficient is evaluated as in equation (1) depending on the load on water (that simulates the real condition of load, for every speed) Δ, the specific weight of water w and the maximum beam b of the hull [2].

$$C_{\Delta} = \frac{\Delta}{w \cdot b^3} \tag{1}$$

Variables are defined as follows (equations (2)-(3)).

$$w = 10^3 \cdot 9.81 \frac{N}{m^3} \tag{2}$$

$$b = 0.243m \tag{3}$$

For each different C_{Δ} there is a different draft value of the hull. The immersed volume of the body depends, in fact, on the load applied on it. N.A.C.A.'s tests display the C_{Δ} interval from 0.95 to 0.02, corresponding to a decrease in terms of draft value.
In order to reduce the independent variables, it is possible to consider only the trim angle (the best trim), that, for selected speed and loads, gives the minimum resistance. It is possible deduce the best trim by reading the value on a plot obtained by the experimental data collected by N.A.C.A. The plot, from [2], shows the dependence between the best trim and the speed coefficient C_v, with C_{Δ} as parameter. Its values changes by the simulation runnings, one per each velocity, while C_{Δ} is

fixed. The analysis takes into account a single C_Δ value that is 0.6 aiming the comparison between the Team S55's numerical data and the N.A.C.A's experimental ones [2]. Actually, the CFD analysis varies according to the speed coefficient C_v defined in equation (4), where V is the stream velocity and g is the gravitational acceleration.

$$C_v = \frac{V}{\sqrt{g \cdot b}} \tag{4}$$

In the T/N n.635 [2] C_v varies in the interval (1;9).
In the hull point of view, water and air stream join the same velocity.
The water line depends on C_v, thus the height of immersed hull surface has to be evaluated case by case.
When the hull floats on the water surface, the load equals the aircraft weight, like in equation (5).

$$C_\Delta = \frac{\Delta}{w \cdot b^3} = \frac{m \cdot g}{w \cdot b^3} = 1.49 \tag{5}$$

Where m is the the total hydroplane replica mass, defined by (equation (6)).

$$m = 21.37kg \tag{6}$$

The water line leaves 0.0069 m underwater at C_Δ=1.49; this value of draft has been obtained by equalizing the Archimedes's force and weight. When the hull detaches from water, the water line is null as well as the water load. The linear interpolation of this two points on the Cartesian plane load coefficient-water line gives the water line at C_Δ=0.6.

2.1 Pre-processing: geometry clean-up and meshing

Computational Fluid Dynamics requires some geometry changes with respect to the actual CAD project, in order to prevent the generation of invalid mesh elements [9]. No concave areas are allowed, so junctions have to be adapted and trailing edges have to be cut. All unnecessary edges have to be deleted, because they determine the creation of many more small cells in mesh generation. In order to do this, the CAD is imported into ANSA environment, a Beta Cae System's software. Only the external surface is involved to the CFD analysis, since only this part interacts with the fluid. Every inner element has to be deleted and gaps have to be filled, correcting the superficial curve as same as cusp at the joining points between two surfaces (fig. 1).

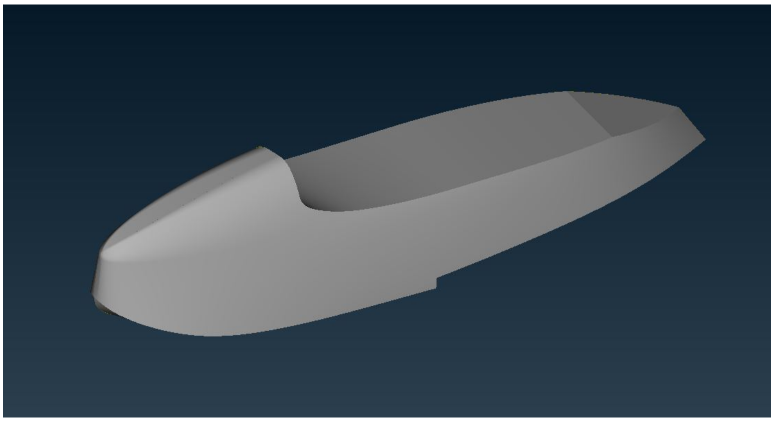


Figure 1 – Clean hull geometry.

While ANSA is involved only in pre-processing phase, the Siemens's CFD software Star CCM+ is used for the meshing phase, the physics conditions setting and the simulation running. . Finally, the software MATLAB is used for the post-processing phase.

The chosen domain is a parallelepiped whose size is based on the hull [1][3][6]. It has been preferred against a semi-spherical one, which has a lower computational cost, because it works better in sub-sonic incompressible simulations. According to CFD requirements, the domain is high ten times the body and long at least 20 times the body. The height and the width of the studied domain are both 14 meters, while the length of the upstream flow is twenty times those of the hull and the length of the downstream is thirty times, to capture completely the wake.
One of the purpose of the analysis is the evaluation of how drag changes with time, during the take off. In order to capture the relative movement among the fluids and the body, another domain is created around the hull. This box corresponds to the overset region centred in the hull, whose size is big enough to incorporate the hull but not too small so as to capture all the body movement. The external domain (the region outside the overset) is the background region.
All regions are meshed using the trimmed mesher to create a volume mesh of cubes extruded from the surface. This mesher realizes a smoother transition than other meshers available between the overset mesh, that moves inside the domain, and the background mesh [3][4][5][7].
The Overset Mesh is an iterative method of *Chimera* type that discretizes a domain through overlapping meshes. When a degree of freedom (DOF) is assigned to the body, the body moves with the overset, because it is fixed in the overset region, in which the governing equations are firstly solved.
The solution is linearly interpolated in the interface between overset and background, hence, a multi refinement step for the mesh is required [7][8].
Furthermore, a finer refinement interests the mesh at the interface between air and water in order to better detect the wake effects and the water surface's behaviour, whereas two progressive box refinements lie around the hull.
Fig. 2 shows the regions in which the domain is divided.

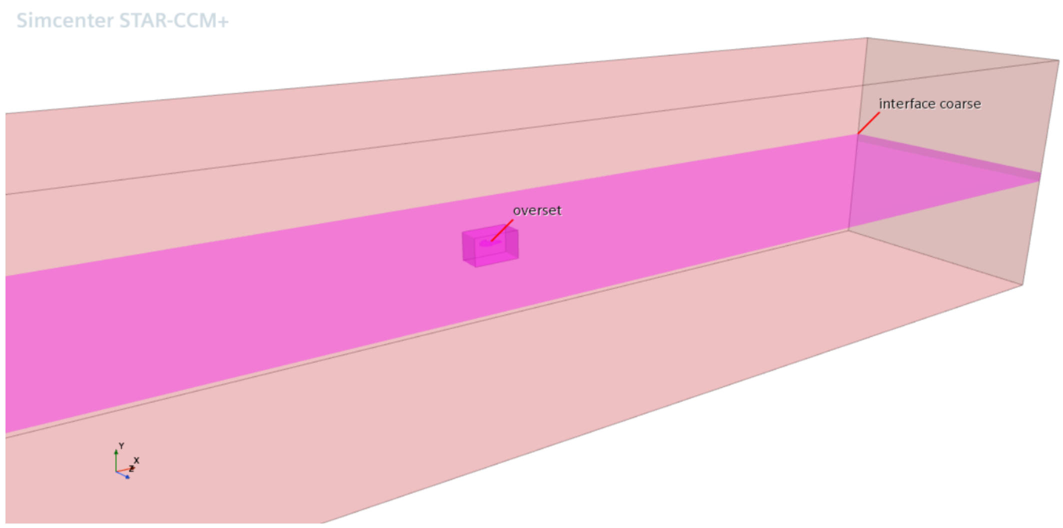


Figure 2 – Regions in the domain.

The final mesh looks like fig. 3:

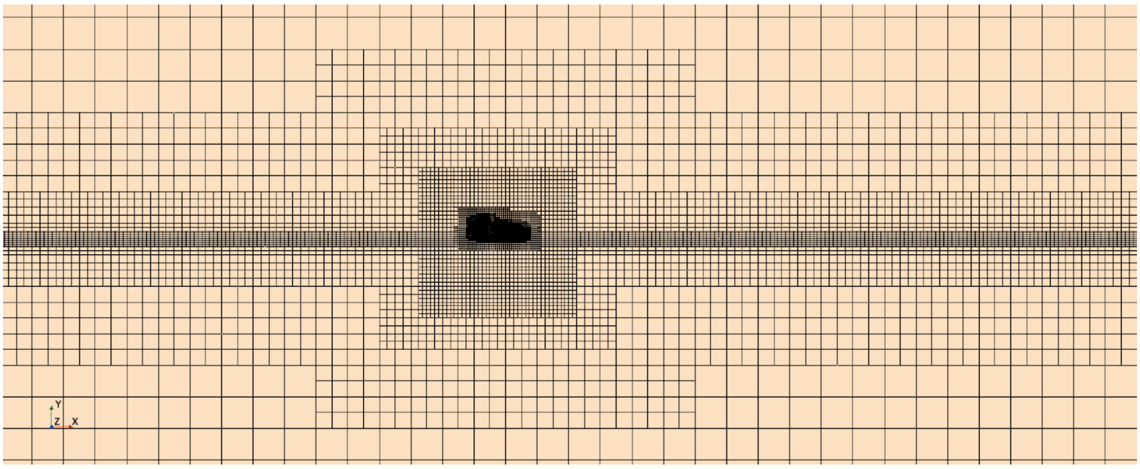


Figure 3 – Domain shape and mesh. It is visible a denser mesh at the fluid interface and in two size box centred in the hull.

The Reynolds number of the system is evaluated as in the equation (7), based on the length of the hull for a speed of 18 $\frac{m}{s}$, and a cinematic viscosity of $1.51 \cdot 10^6 \frac{m^2}{s}$.

$$Re = \frac{U_{inf} \cdot L}{\nu} = 1.44 \cdot 10^6 \tag{7}$$

A set of prism layers is extruded from the surface of the hull to better analyze the behavior of the boundary layer between the surface of the aircraft and the external flow. Hence, the total thickness of the boundary layer is evaluated knowing the Reynolds number. The other parameters to define are the stretching factor, set to 1.2, and the height of the first cell centroid starting from the wall, calculated to obtain a y^+ equal to 50, placing it in the logarithmic profile region. These are the conditions way ten prism layers are extruded from the surface (fig. 4).

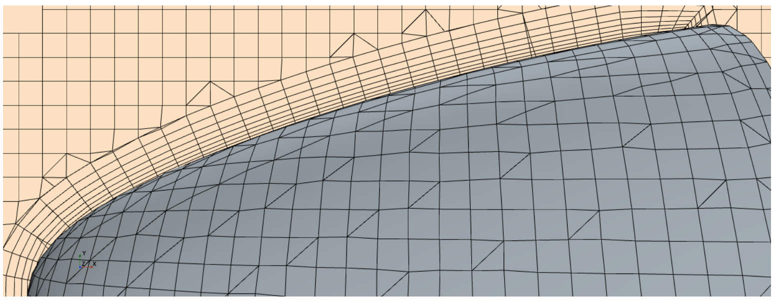


Figure 4 – Prism Layers

2.2 Physical models

The fluid field is modeled as a three-dimensional mixture composed by two ideal Eulerian phases, a liquid one (water) and a gaseous one (air). The equation of state considers two different sets of property, one per fluid. Air and water do not mix during the simulation run, neither through the overset running, and no phase change occurs during the simulation. The relative wind and water flux move at the same velocity.

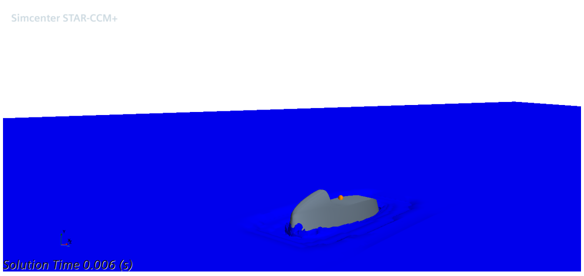
The Volume Of Fluid (VOF) is a multiphase model suitable to predict how the interface of immiscible fluid mixtures changes position during the time, when the contact area is relatively small with respect to the single phase extension [7]. The properties of each phase, such as density and dynamic viscosity, have to be specified. Physical quantities are defined for every single cell by volume fractions α; of course, at the interface it has value that varies between 0 and 1, and the sum of volume fractions for each cell has to be equal to 1 (equation (8)).

$$\sum \alpha_i = 1 \tag{8}$$

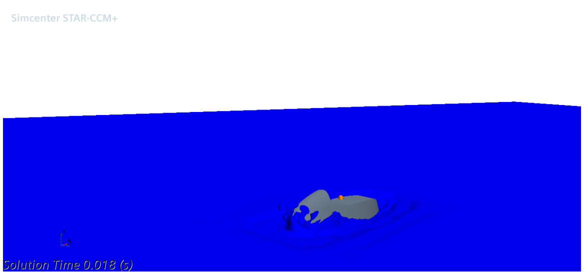
The sum condition allows the solver to calculate the transport equation for α for only one phase. The scheme selected to discretize convective terms is the High-Resolution Interface-Capturing (HRIC), a second order accurate scheme, more precise but less stable than the First Order solver. HRIC enforces strict conditions about CFL, set as 0.3. CFL is a parameter that enstablishes the time-step in use for the simulation (equation (9));

$$\Delta t = \frac{\Delta x}{U_{inf}} \cdot CFL = 1.5 \cdot 10^{-4} s \tag{9}$$

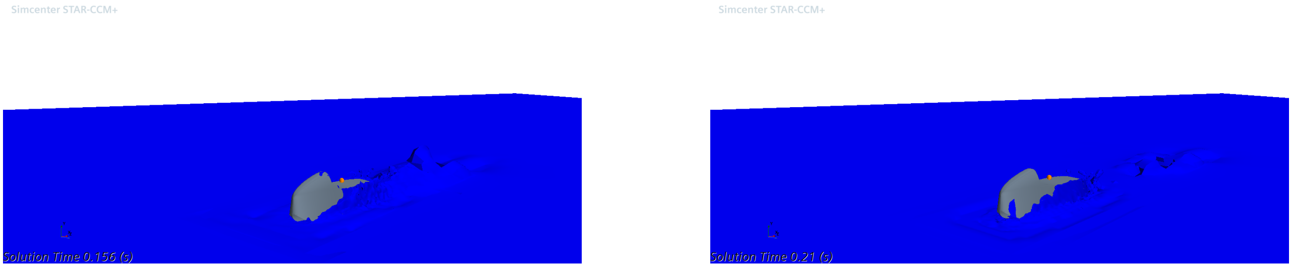
The parameter Δx represents the cell size near to the hull.
A good way to simulate motions is introducing the Adaptive time-step and the Adaptive Mesh Refinement. The Adaptive time-step adjusts time step in order to keep the CFL on the target value, while the Adaptive Mesh Refinement regenerates the mesh during iterations splitting edges to generate new cells and merging others. However, since the mesh refinement around the hull and in the interface between the two phases has been dense enough, it has been made the choice not to use the two models.
Volume fractions provide the spatial distribution of each phase at a given time. The segregated flow fits simulation having low Mach number, hence incompressible flows. It uses gradient method to calculate variables at the cell faces. Viscous effects are fundamental to simulate properly the drag, but the settings of the physics must think also of analytical model request. Reynolds number suggests the need to include turbulent models, being this number much superior than the indicative target figure for a transition between a laminar to a turbulent boundary layer. Unsteady Reynolds-Averaged Navier-Stokes's equations (URANS) govern turbulent models.
K-Epsilon model is selected as the most appropriate URANS model. The Eulerian Multiphase model uses an individual K-Epsilon turbulence model for each Eulerian phase.
The presence of periodical phenomena, such as the wake generated by the body, requires an implicit unsteady simulation to capture the behavior of the aerodynamic interactions and to have a precise result.
Having two fluids at different densities, gravity effects are important: gravity has to be involved for a correct interaction between air and water, and to rightly evaluate the hydro-static pressure. Gravity concerns also the interaction between the body and the fluid. However, the body geometry is empty: weight must be specified as the whole mass lies in the centre of mass.
The surface of water is modelled as a constant flat wave running over the keel; a flat wave behaves as calm water until the flow bumps an obstacle, then the wake shows waves. An appropriate mesh refinement can catch the phenomenon (fig. 5 a-d). The wave is created through the VOF Waves model, producing a field function based on the free water surface. VOF waves is a special case of VOF model. It can also simulate surface gravity waves on the interface between a light fluid and a heavy fluid, as air and water. When the wave reaches the step at the end of the forebody, the wave displaces water away from the hull, the flux detaches and the hydro-dynamic drag quickly decreases. Wave height increases proportionally to the velocity.



(a) t=0.006 s



(b) t=0.018 s



(c) t=0.156 s (d) t=0.21 s

Figure 5 – Wave progression in the wake.

For rigid bodies such as the hull, modeling the motion of the center of mass is sufficient to obtain the total movement. Besides, the relative motion of any other part of the body can be extrapolated from t0his center of mass, considering that the relative distance between internal points does not change by definition.
The application of motions happens thanks to a dedicated Star CCM+ model, the Dynamic Fluid Body Interaction (DFBI) [7].
The DFBI emulates the motion of a rigid body in response to the fluid forces. The DFBI can compute the resultant force and moment on the body, and it solves the equations of motion to find the new position of the rigid body. It is necessary to define a local Cartesian reference frame and to know the moments of inertia of the body about a fixed reference point, being usually the center of mass. The 6-DOF motion solver is carried creating the DFBI motion object and granting it to the body.
The hull does not have lifting surfaces, thus the lift has to be added as an external force applied on the hull. Lift has been calculated considering a load contingency factor for the take off equal to 1.09. Also weight is applied as external force.
The simulation carried out takes advantage of the DFBI solver, but different DOF are left free to move, according to the type of simulation. At first the DFBI is set to have only 2 DOF free to move, vertical translation and pitch motion. Later on it has been decided to leave only the vertical translation free to move, due to a stability issue: the absence of control surfaces in the hull does not allow the attitude control. Besides, since each DOF adds up to the total number of calculations in the simulation, it means that the greater the number of DOF are left free to move, the greater the computational cost will be.

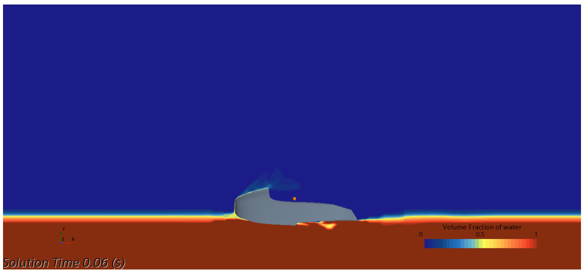
2.3 Initial conditions and boundary conditions

Solving the URANS requires specifying the values of main quantities at initial time, in terms of pressure, velocity, turbulence parameters, and volume fractions of the phases to compute the govern equations of the fluids.
The take-off takes place at sea level, thus the initial air pressure coincides to the reference value (1 atm). The hydrostatic pressure follows Stevin's law, which is a linear law. The initial velocity corresponds to the take off velocity.
The setting of turbulence requires the initial intensity and the length scale. The length scale corresponds to 5% of the characteristic length of the problem, that is the hull transversal width. As regards turbulence intensity a value of 1% has been chosen. Referring to the stream direction, all the surfaces are defined as velocity inlets except the rear surface, which is pressure outlet. In other words, the flux can enter the domain from every direction except the back, from which it can simply exit [1][7][8].
The turbulence parameters and the volume fraction are the same as initial conditions.

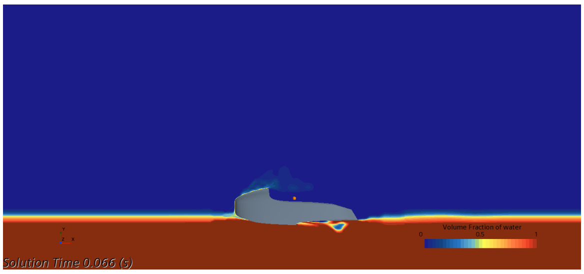
3. Simulation and results

The simulation is executed at velocity equal to 18 m/s, one DOF free to move (the translational one), and an overall physical time of one second. It is important to remark that the analysis returns a lower take-off time than the real one, because the simulated velocity is constant at the detachment speed.
It has given following results:

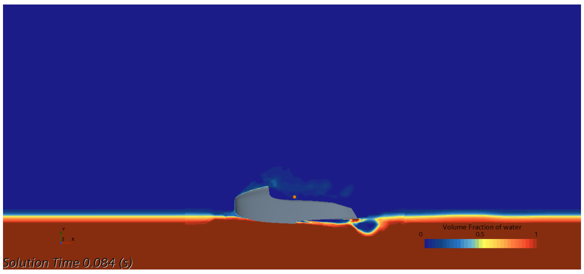
(a) t=0.036 s



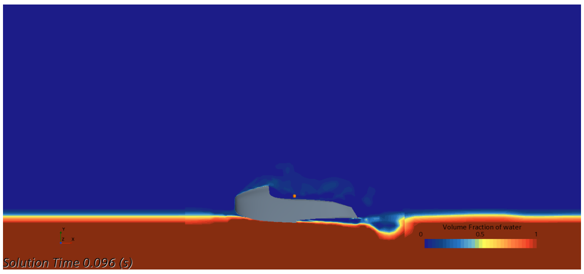
(b) t=0.06 s



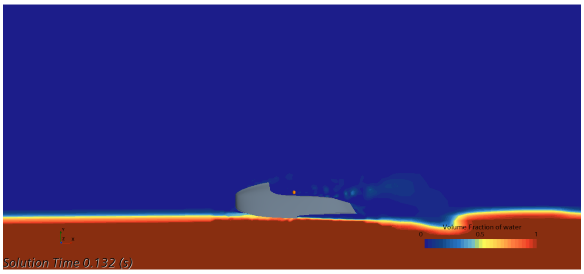
(c) t=0.066 s



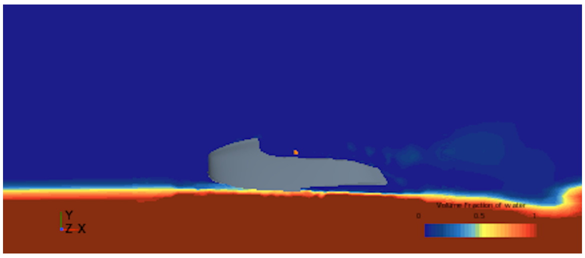
(d) t=0.084 s



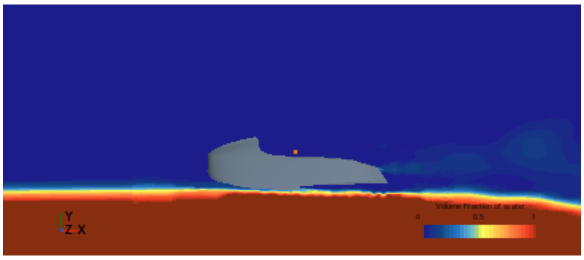
(e) t=0.096 s



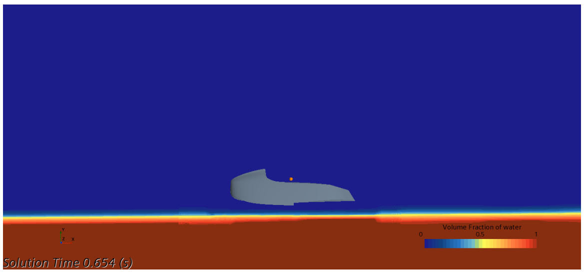
(f) t=0.132 s



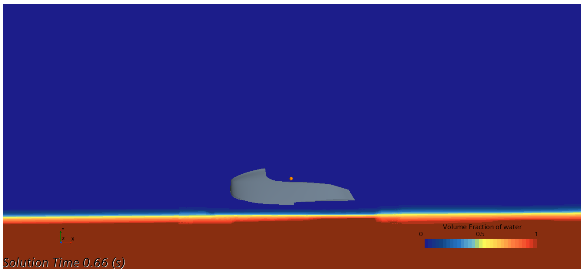
(g) t=0.144 s



(h) t=0.168 s



(i) t=0.654 s



(j) t=0.666 s

Figure 6 – Take-off transition.

The volume fraction of water is represented in fig. 6 and it is defined for each cell as the ratio between volume of water and total volume of the cell. The trend of the field function during the time shows the movement of the interface between water and air. It is clearly visible the vertical movement of the hull, due to the lift applied on it, and the wave coming off the wake. The fig. 6.b shows the presence of a thin air layer right after the step of the hull. It is going to evolve into a bubble a few instants later as seen in the subsequent steps. The presence of the bubble is to be expected as shown in the N.A.C.A. experiments [2] and it validates the assumption that the step on the hull was designed to reduce drag resistance thanks to the bubble that it generates. This statement is confirmed by the evolution of the drag plot shown in fig. 7

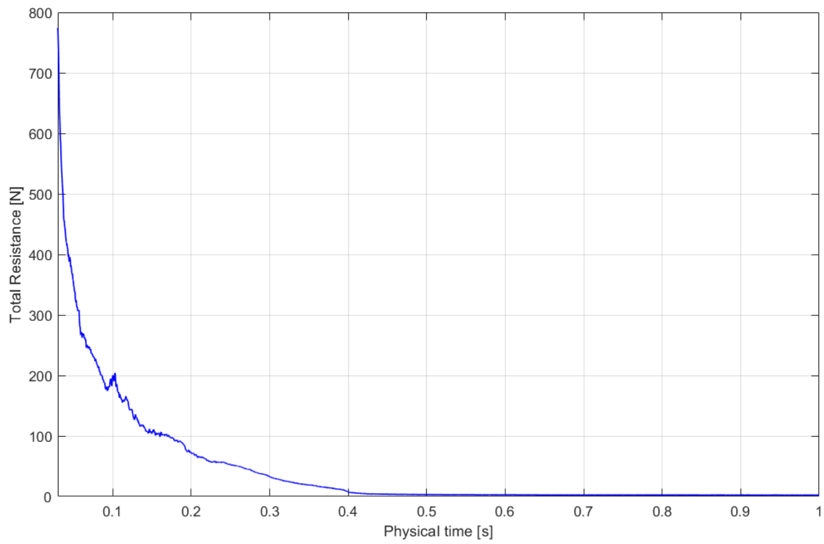


Figure 7 – Resistance plot in function of time

The rapid decrease of the resistance, especially up to 0.4 seconds, is caused by the formation of the bubble near the step, as mentioned before. Finally, the fig. 8 shows the vertical motion of the hull in terms of distance covered by the center of mass of the hull:

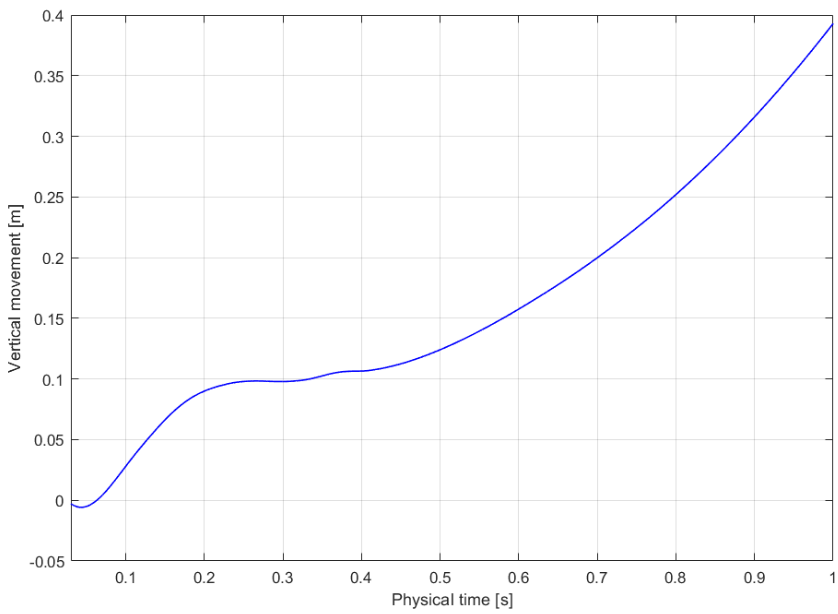


Figure 8 – Vertical translation

4. Experimental data comparison

The second set of simulations has been performed with same DOF left free to move as the first one. This condition matches those of the N.A.C.A's tank test, with external load and trim applied and the test were performed at different speed [2].

In the tank test, a 1:5.25 scaled hull of S-55-X is suspended thanks to a designated structure. The structure applies the load on the water, through the hull. The dead rise of the afterbody increases with distance after of the step, giving a wind in the bottom surface.
The test follows the general method in which the experiments includes a number of constant speed runs. The hull, in the simulations, is constantly set at best trim, corresponding to the minimum resistance for selected speeds and loads. The angle of attack of the hydrofoil, with the task to provide lift to simulate a realistic condition, was adjusted to make the model take off approximately at 14.78 m/s.
Thrust is not involved in the test since its moment would reduce the estimated maximum positive moments in the real case. In order to make a comparison between the CFD analysis and the experimental data obtained by N.A.C.A, the speed coefficient's interval is chosen in order to matching the hump speed range. The hump speed is the velocity at which the hull starts planing on water surface and consequently resistance drops [13]. The Team S55 hull's hump speed is equal to equation(10):

$$V_h = T \cdot \sqrt{L} = 4.51kts = 2.32m/s \tag{10}$$

L is the reference hull's length and it is equal to 2.29 ft and T is a coefficient whose value is 3 for planing hulls (such as S55-X's one). The speed coefficient relative to the hump speed is equal to 1.54, hence, the others C_v values have been chosen in order to have an equally spaced interval. The analysis, thus, considers five C_v values, for a C_Δ fixed, in order to reduce computational cost by obtaining only the curve $C_R - C_V$ for $C_\Delta = 0.6$, in the interval (1;2) as in tab. (1).

C_v	$V[m/s]$
1	1.54
1.27	1.96
1.54	2.32
1.77	2.73
2	3.09

Table 1 – Velocity values considered in the comparison.

The load applied on the water by the hull (Δ), corresponding to $C_\Delta = 0.6$, is equal to 84.46 N. An external force that is equal to the difference between Δ and hull's weight has been added to the body to reach the desired C_Δ condition. For each analysis the time averaged values of resistance and trimming moment, referred to center of mass, have been calculated and, consequently, the relative coefficients. The results are shown in figures (9) and (10).
The CFD trend can be compared with the experimental N.A.C.A's one. The curves show the same slopes until the hump speed ($C_v = 1.54$); Team S55's resistance coefficient is higher than the experimental value because of the made approximations. Firstly, the initial dead rise value comes from a linear interpolation, thus it generates a discrepancy with respect to the real case. Then, the replica best trim fixed for each simulation is taken from the N.A.C.A.'s plot [2], that considers a different model size, and it could not be the same as N.A.C.A. without further considerations. Also a finer mesh could reach more precise values, but it has been avoided because of the eventual prohibitive computational cost.
Even if the curves are not exactly the same, the results are satisfying: they show the same coefficient resistance trend in increasing and then decreasing, as fig.9 displays; the eventual differences depend on the interpolation.
The fig. 10 shows the C_m coefficient as a function of C_V. The mismatch between the curves is caused by different C_R values among CFD results and experimental ones.

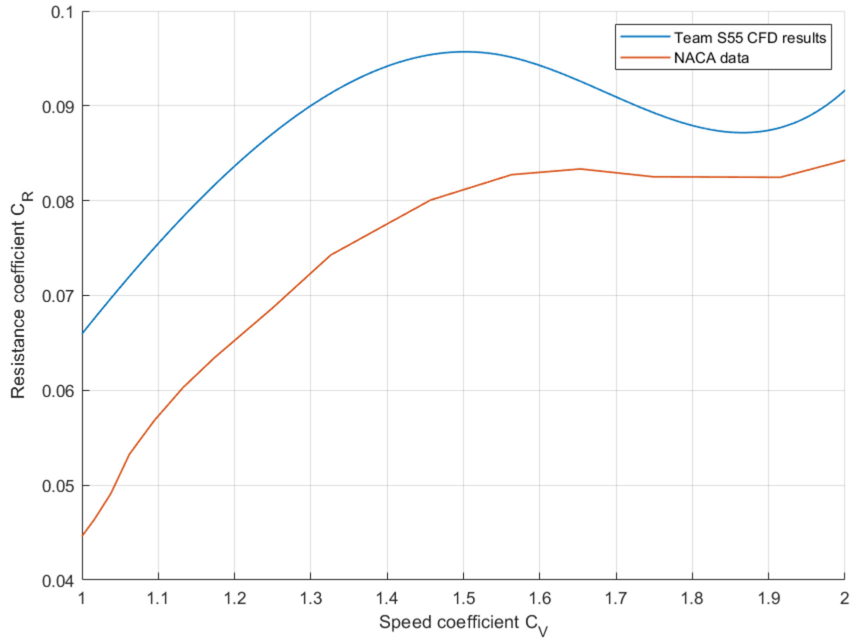


Figure 9 – Resistance Coefficient plot

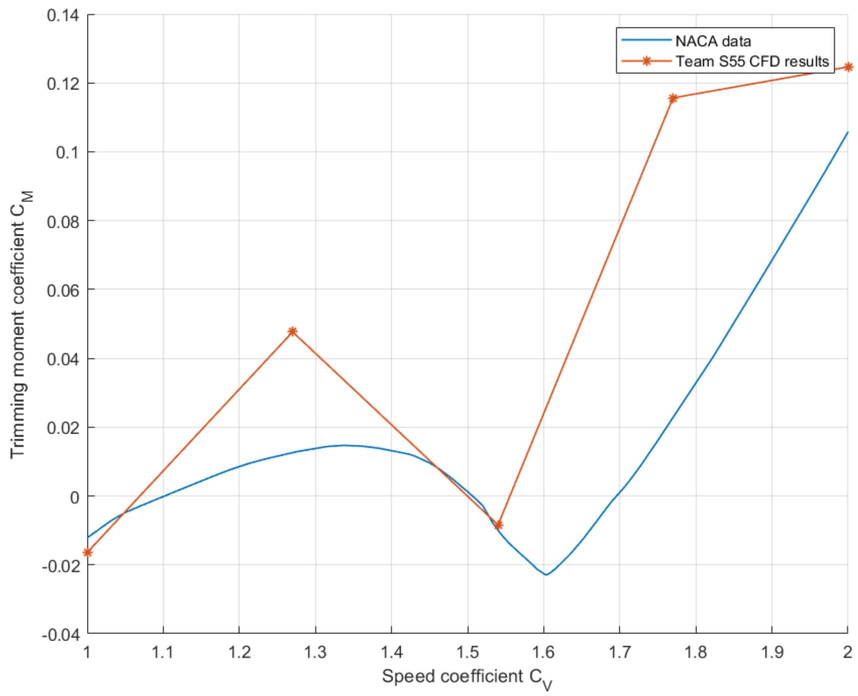


Figure 10 – Trimming moment coefficient plot

5. Conclusions

Knowing the flux properties, the load and the trim, it is possible to realize a realistic CFD take-off simulation. The complexity of the problem has not obstructed the correct running of the simulations. In fact, the analysis converges and the hull takes off properly.

The Aerodynamics Section of the Team S55 has been able to model the take-off, considering the involved forces and moments. In the first simulation set the total resistance decreases as expected, due to the air bubble generating behind the step, and in the latter set of the analysis has been pointed out the extraordinary hump region's behaviour of planing hulls, as the S55-X's one. Furthermore, a group of students has been able to reproduce the same experimental conditions in N.A.C.A's tests, by obtaining comparable results with the N.A.C.A's ones.

6. Acknowledgments

The authors would like to thank the companies and people who contributed to this paper and to the 1:8 scale replica of the S55X. First of all, the entire Team S55, made up of young students full of enthusiasm, then Professor Enrico Cestino and Engineer Vito Sapienza, who constantly support the project by making their experience and time available to the team. We would also like to thank the Politecnico di Torino which has been supporting the team financially since 2017 investing in our projects. Last but not least, thanks to all the companies and organizations that collaborate with the Team S55: Replica 55, Ellena-SPEM, BETA CAE Systems, Altair Engineering, HPC Polito, Siemens, Mike Compositi, Ades Group, MODLab Design and Erre ti Compositi. In the end, a special thank is to our colleague Riccardo Piani, for the time he has spent on us and its precious advice.

7. Contact Author Email Address

Please contact:

Team S55: teams55.polito@gmail.com
Marika Sinisi: marika.sinisi@studenti.polito.it

8. Copyright Statement

References

[1] Yang Guo, Dongli Ma, Muqing Yang and Xiang'an Liu. Numerical analysis of the take-off performance of a seaplane in calm water *Applied Science*.

[2] John M. Allison. Tank test of a model of one hull of the Savoia S-5-X Flying Boat - N.A.C.A. Model 46. NACA T/N no.635.

[3] Baldon C., Indelicato R., Bottino N., Sinisi M., Carrone F., Cantanna G., Cestino E., Sapienza V. S55 Project: CFD Analysis of an Historical Seaplane. *32nd Congress of the International Council of the Aeronautical Science*, Shanghai, China. September 2021.

[4] Matthew W. Floros, Jayanarayanan Sitaraman. Parallel Unsteady Overset Mesh Methodology for Adaptive and MovingGrids with Multiple Solvers. *Nato Unclassified +SWE+AUS*. RTO-MP-AVT-168. Hampton, Virginia.

[5] Rajiv Shenoy and Marilyn J. Smith and Michael A. Park. Unstructured Overset Mesh Adaptation with Turbulence Modeling for Unsteady Aerodynamic Interactions. *Journal of Aircraft*, Vol. 51, No. 1, January–February 2014

[6] Pietro Casalone, Oronzo Dell'Edera, Beatrice Fenu, Giuseppe Giorgi, Sergej Antonello Sirigu and Giuliana Mattiazzo. Unsteady RANS CFD Simulations of Sailboat's Hull and Comparison with Full-Scale Test. *Journal of Marine Science Engineering*, Vol. 8, No. 394, May 2020.

[7] *Star CCM+ 2020.1 Documentation file*. Siemens Manual, 2020.

[8] *Ansa for CFD Brief User Guide*. Beta CAE System, 2020.

[9] Cestino E., Sapienza V., Frulla G., Pinto P., Rizzi F., Zaramella F., Banfi D.. Replica 55 Project: A wood seaplane in the era of composite materials. *31st Congress of the International Council of the Aeronautical Science*, Belo Horizonte, Brazil. September 2018.

[10] F. Valpiani, A. Polla, P. Cicolini, G. Grilli, E. Cestino, V. Sapienza. Early numerical evaluation of fluid structure interaction of a symple wedge geometry with different deadrise. *Italian Association of Aeronautics and Astronautics XXVI International Congress*, Pisa, Italy. September 2021.

[11] Nicolosi, Valpiani, Grilli, Saponaro Piacente, Di Ianni, Cestino, Sapienza, Polla, Piana. Design of a vertical ditching test. *32nd Congress of the International Council of the Aeronautical Science*, Shanghai, China. September 2021.

[12] Favalli, Ferrara, Patuelli, Polla, Scarso, Tomasello. Replica 55 project: aerodynamic and FEM analysis of a wooden seaplane. *31st Congress of the International Council of the Aeronautical Science*, Belo Horizonte, Brazil. September 2018.

[13] Hump speed water or go to the next URL `https://www.merriam-webster.com/dictionary/hump$%$20speed#:~:text=Definition$%$20of$%$20hump$%$20speed,water$%$20resistance$%$20reaches$%$20a$%$20maximum`