CFD Wind Tunnel Assessment on the case study of the MOTHIF Blown Flap

Paolo D’Alesio1, Giorgio Travostino1, Philippe Planquart2 and Gertjan Glabeke2

1 Piaggio Aerospace, Aerodynamic Department, Villanova d’Albenga (SV), Italy
2 Von Karman Institute, Sint Genesius Rhode, Belgium

Abstract. The increasing interest in Small Air Transportation (SAT), to enhance global connectivity, is highlighting the need for reducing take-off distance: lift coefficient has to be increased without penalizing the configuration drag, weight and complexity. Within the EU Clean Sky 2 projects, a blown flap configuration has been developed to allow STOL capabilities of a future affordable and green commuter belonging to EASA CS 23. The blowing system design choice is aimed at keeping the inevitable associated increase in pitching moment very low, and not penalizing performance when blowing fails. In collaboration between Piaggio Aerospace and the MOTHIF consortium, a wing model was designed, built and tested in the VKI’s large subsonic L1-A wind tunnel, reproducing bi-dimensional preliminary results obtained during the design phase. In order to reduce and predict three dimensional and blockage effects, CFD has been used extensively to obtain a proper test chamber configuration and to reproduce some wind tunnel test results, so that both the accuracy of wind tunnel and design methodology can be assessed. This work presents the comparison between experimental data and CFD simulations to demonstrate the usefulness of simulations to reduce both the risk for erroneous results in experimental activities and their related cost.

1. INTRODUCTION
SAT aircrafts development, capable to take off from the most remote areas, with a runway shorter than 800m, is crucial to meet the European FlightPath 2050 “d2d 4h” connectivity goal.

The aircraft should have a high lift system capable of generating high lift coefficients but without penalizing the overall drag, the aircraft weight and the trim drag, i.e. shorter take off distances. Therefore, a new high lift concept is needed.

The design process for every new concept, after the theoretical design, should foresee some experimental campaigns, to assess the expected gains of the proposed solution.

The experimental campaign allows the engineer not only to validate its design process and the expected outcomes but also to assess the effectiveness of its design methodology and tools.

Experimental data are then transformed into performance data, proving the validity and advantages of the proposed solution.

Whatever the outcome of the experimental campaign, it should never be considered as a failure: even a “don’t-know-how”, the need for more accurate tools or the need to completely change the design perspective are useful results of a test campaign, setting the right path for future innovation.

It is clear how the experimental campaign accuracy, its ability to reproduce the test correctly, its limitations and many other related aspects are crucial for a correct critical review of the results.

Thanks to CFD, by controlling and predicting the experimental results, issues and limitations, it is not a specific design that can be validated, but a whole design methodology.
Furthermore, the chance to deeply and critically analyze the test method and experimental facility arrangement provides the opportunity to test rapidly new configurations, so reducing experimental costs.

Concerning a wind tunnel test campaign, CFD can be extensively used to take care of two main issues:

- the test chamber arrangement, i.e. side walls, wind tunnel wall lips, balance sting, specimen supports, etc. are always affecting the flow around the specimen
- The specimen dimensions, i.e. results can be affected by blockage and for which the simultaneous similarity of the Reynolds, Mach and Froude numbers requires expensive tests in cryogenic and pressurized wind tunnel.

Nowadays, it is always a trade-off between model scaling and test flow speed on the one hand, and to choose of which flow characteristic is more important for the experiment itself, i.e. whether compressibility is of fundamental importance or whether the flow inertia is enough.

Simulations can instead help in validating the results obtained in the design phase through wind tunnel testing, even if they are obtained with a high blockage ratio, i.e. large model but relatively small test chamber.

This may open up the possibility of also using smaller (related to the size of the specimen) wind tunnel installations, since small geometrical change with respect to the main geometric wing features may no more need large wind tunnel test chamber. This may result in increasing the pace of innovation and increasing speed and accuracy of configuration tests and also reducing experimental costs.

This article is precisely intended to share the experience of what can be called a “Critical Test Design Review” (CTDR) in a wind tunnel test campaign and the crucial role of CFD.

The wind tunnel test campaign was conducted within the Clean Sky 2 AIR ITD framework: the objective is to test a blowing flap model reproducing bi-dimensional data obtained in the design phase and demonstrating the advantages of this new high lift device

The specimen was designed and built by Piaggio in close collaboration with the MOTHIF consortium composed by SONACA and VKI (von Karman Institute). The Belgian manufacturer SONACA supplies the flap, main airfoil and side plates, allowing for different flap deflections and gap positions, while Piaggio Aerospace develops the trailing edge module including the inner blowing piping, built with SLS ALM manufacturing.

In the following chapters we will illustrate the capabilities of CFD to predict flow behaviour for specific test chamber arrangements and how the arrangement itself influences the expected results.

Moreover, we will present the use of CFD to re-assess wind tunnel results with high blockage ratio. This, is to validate the code and results in the design phase, pointing out the advantages and disadvantages of the large-scale use of CFD.

2. EXPERIMENTAL ARRANGEMENT: WIND TUNNEL DESCRIPTION
The selected wind tunnel is the VKI low speed wind tunnel L1-A, which has an open jet test section of 3m diameter and 4.5m length. Its maximum speed can vary continuously from 0 to 60 m/s. The contraction ratio is 4 with a typical turbulence level of 0.3%.

The wind tunnel freestream tests speeds are 20m/s, 30m/s, 35m/s and 42.5m/s, to evaluate blockage and speed influence on aerodynamic behaviour. Different jet blowing pressure (and so blowing speeds) are also tested.

Figure 1. VKI L1-A wind tunnel (red circle: model location)
3. GEOMETRIC MODEL and MESH

Following the guidelines for the new high lift devices, outlined in the introduction, a blowing flap has been designed, combining the Fowler flap with the blown jet, to obtain also good blow off performance.

To assess the validity of the design simulation data, the collaboration between Piaggio and SONACA produced a model consisting of the main airfoil, the inner blowing system and the flap.

This model has the following characteristics:
- 1.5 m of span
- 1.35 m of chord with retracted flap
- 0.181m maximum airfoil thickness (13.4% of the chord)

The wind tunnel geometry with the specimen inside the test chamber, thus the geometry subject of this article, is illustrated below. Large side walls have been added in the wind tunnel to avoid 3D effects.

Taking advantage of symmetry, only half geometry is used for the simulations. To consider high blockage conditions, flap=30° (landing configuration) and AoA=4° and 8° have been simulated.

While assessing the test chamber arrangement, the model geometry is modified based on the CFD results, to finally obtain a satisfactory test chamber setup, which is then maintained.

The mesh is obtained with ICEM and it is a 70 million cells hybrid mesh, distributed following a “matrioska paradigm”, with an extensive application of higher density cells. Particular attention was paid to the discretization of the suction region of the airfoil, because of the higher gradients developed.
4. SIMULATION SETUP

Simulation CFD software has been Metacomp CFD++ [1].

Wind tunnel speed is 42.5 m/s, to reproduce high blockage. This condition is imposed considering wind tunnel inlet as a total pressure reservoir, since the open test chamber is at ambient pressure, imposed as characteristic based. The other elements are walls, except for the symmetry plane.

Two different turbulence models, are used: both k-ω SST and k-ε, initialized at WT turbulence level.

Simulations also considered blown flap configurations: the blowing flow is imposed at the inlet of the inner blowing piping with a mass flow boundary condition, deriving separately from 2D simulations.

Unless otherwise specified, all the simulations are fully second order starting from the first iteration.

5. RESULTS: TEST CHAMBER ARRANGEMENT SETUP – NUMERICAL OPTIMIZATION

CFD has been helpful in designing a correct wind tunnel test chamber arrangement: its capability of predicting vorticity and Cp distribution has been important to define a correct endplate positioning.

Closed roof and bottom open side with pillars characterized the first WT arrangement. The CFD simulation visualizations using y-vorticity components shows an upward flow shift causing a loss of the specimen aerodynamic angle of attack and consequently a loss in lift. The y-vorticity development was enforced by the pillars structure of the specimen, but it still arises if the pillar structures were faded, as highlighted on the right below pictures, so proving that the wind tunnel inlet lips are producing it.

The y-vorticity flow tube locking caused also half of the wind tunnel outlet tube to be at very low speed, with a small recirculation on the bottom lip: these effects have been counter-checked and found experimentally using tuft visualizations. The corrective action consisted in placing a bottom endplate:

![Figure 7. First WT Chamber arrangement: y-vorticity.](image1)

![Figure 8. Effect of pillars absence on y-vorticity.](image2)

![Figure 9. Y-vorticity with the bottom endplate.](image3)

![Figure 10. Cp distribution over the airfoil (symmetry plane): first arrangement (orange) vs bottom endplate adding (blue.)](image4)
The bottom endplate forces the flow straightness and increases the lip vorticity diffusion: this leads to a recovery of the incidence and the suction development on the airfoil, as plotted in Fig. 9 and Fig. 10. A further step forward was the addition of an opening on the roof endplate, to increase the pressure on the airfoil, by reducing the flow acceleration in the test chamber on the upper side.

The table below shows the increase in specimen CL, at alpha=8° and flap=30°, thanks to the WT chamber development and refinement: the corrective actions recovers the deficit developed by the wind tunnel test chamber arrangement, passing from 47% of lift loss (first test chamber configuration) to 19% of lift loss (last chamber refinement) with respect to free air simulations (CL=3.52).

| Flap=30°, alpha=8°          | Cl   |
|-----------------------------|------|
| No bottom plate, rooftop closed (Figure 7) | 1.87 |
| Bottom Plate, Rooftop closed (Figure 11)     | 2.63 |
| Bottom Plate and Hole on Rooftop (Figure 12) | 2.86 |

6. RESULTS: BLOW-OFF LANDING CONFIGURATION RESULTS
At AoA=8°, CFD has been capable to reproduce the resulting CL from wind tunnel experiments at flap=30°. Moreover Cp distribution over the specimen is in good agreement with experimental results:

Figure 11. Cp distribution in the test chamber with bottom endplate.

Figure 12. Cp distribution in the test chamber with a hole in roof endplate.

Figure 13. Cp distribution comparison at AoA=8°: CFD (blue circles) vs WTT (orange squares).
This means that also Cmy is in very good agreement between simulations and WTT results. Looking at the coefficients:

Table 2. Comparison between CFD and WTT for flap=30° and AoA=4° & 8°.

| AoA | C_L | C_mmy | C_L | C_mmy | Δ C_L [%] | Δ C_m [%] |
|-----|-----|--------|-----|--------|-----------|-----------|
| 4°  | 2.68| -0.75  | 2.626| -0.7   | +2%       | +7%       |
| 8°  | 2.86| -0.625 | 2.892| -0.66  | -1.2%     | -4.3%     |

The values shows that the CFD, using k-ω SST turbulence model, is in very good agreement with WTT, for blow off conditions, although the above absolute results are smaller than those in free air. When using k-ε, with the same mesh and boundary conditions, CFD cannot sustain adverse pressure gradient rearward the airfoil, causing separation on the trailing edge.

K-ω SST, instead, can deal better with the adverse pressure gradient and results in a large recirculation on the lower rear section of the side endplate, whilst the flow on the airfoil remains attached.

7. RESULTS: BLOW ON SIMULATION

Blowing conditions during wind tunnel tests results in high gradients, as proven by the growing recirculation on the side endplate: these are located on the lower rear section of the side endplate growing bigger when, even alternatively, incidence, flap deflection and blowing speed are increasing.

The CFD simulations in “blowing-on” did not converged even if using the same mesh and k-ω SST turbulence model deriving from blow-off simulations, proven to be in line with WTT. Also, a different simulation strategy initializing CFD with first-order equations then passing to fully second-order formulation, has been tested: large separation starting from endplate still occurs.

Instead, the blowing jets in terms of outlet speed and mass flow are in good agreement with WTT.

As explained previously, blowing-on makes the adverse gradient higher. It can be supposed the mesh should be finer, in order to correctly simulate the evolution of the flow characteristic.

Trying to have a smaller gradient to substantiate the above hypothesis, a simulation at alpha=4° and flap=15° whilst blowing, has been done, with the current used mesh. Different blowing speed have been tested and CFD computations show again a very good agreement with WTT data, with a margin on CL of 10%, (WT CL=2.25; CFD CL=2.03). So, the hypothesis is proven.
Figure 16. Cp distribution comparison at AoA=4°, flap=15°, Blow On at Press Ratio=1.48: CFD (blue circles) vs WTT (red circles).

Figure 17. WT PIV and CFD speed isocurves comparison, for AoA=4°, flap=15°, Blow on at Press Ratio=1.48: CFD (black) vs WT (red).

Speed isocurves comparison between WT PIV and CFD shows a good agreement for the Blow On simulations: the lack in speed resolution for CFD and the difference in the isocurves slopes is due to the gradient resolution and it is directly correlated to the CFD Cp underestimation: still a finer mesh is needed to have the same accuracy as for the “Blow-Off” simulation.

It is important to underline that PIV in this case is misleading for the jet region, since that region is not seeded: thus, particles present are entrained by the jet, so, having a chaotic behaviour, as inferable by the PIV standard deviation and uncertainty values.

Figure 18. PIV calculation uncertainty value on the trailing edge, Blow On conditions.

8. CONCLUSIONS
A blown flap specimen has been manufactured by Piaggio and SONACA and then tested, to assess the performance of a new high lift device concept. The experimental campaign has been conducted in the VKI subsonic wind tunnel, providing results for both Blow-Off and Blow-On conditions.

The present paper aims to demonstrate the usefulness of CFD for a comprehensive CTDR (Critical Test Design Review), capable of improving the accuracy of experimental tests, assessing design tools and methodology, but also highlighting their respective limitations.

CFD simulations have made it possible to determine the correct arrangement of the wind tunnel test chamber, by quickly simulating different configurations.

The simulations prove the undesirable impact of important flow features on the specimen lift and suggest the positioning of end plates to obtain a proper flow field. Once an adequate wind tunnel test chamber has been obtained, high blockage wind tunnel results (higher flap deflection, incidence =4° and 8°) has been calculated through CFD simulation, resulting in a good agreement in terms of lift value and Cp distribution, thus also in terms of pitching moment. On the other hand, in blowing conditions, gradients are higher and the difference between CFD and WTT is higher, but still inside 10%: mesh should be finer to better take care of the gradient increase, so that the simulation can be reliable.

In this way, the use of CFD for the preparation and re-evaluation of wind tunnel tests allows the engineer to predict and control the experimental results, also pointing out the limits of the test itself.
More in general, the blockage is no longer just a troubling WTT side effect, but something that can be predicted and controlled, becoming one of the expected outcome from the design phase that must be validated experimentally. One of the main outcomes of the presented work is the demonstration that CFD can predict both the blockage effect that occurred in WTT and the effectiveness of the changes to the test chamber arrangements to guarantee the desired level of experimental accuracy.

Having proven the simulation accuracy by comparing CFD with wind tunnel flow visualization and test results data, the tools, the methodology and the skills used in the design phase are consequently validated. CTDR gives consistency to the results and methodology used in the design phase and to the predicted performance benefits of the proposed configuration evaluated in free air.

Through CTDR the blowing flap system performance predicted during the design phase, promising to let the aircraft achieve the FlightPath 2050 SAT connectivity goal, so 800 m runway target for take-off operations, is substantiated: the predicted performance, at maximum flap deflection and blowing pressure ratio, of reaching the 86% more of the CL target can be considered valid and achievable.

Crucial is the engineering skill in simulating and reading CFD results: still this approach is highly tailored but makes the difference design “know how”.

K-ω SST model has been proven to be the most effective turbulence model in solving the above flow characteristics, but its accuracy is strictly dependant on the mesh discretization, which is correlated to the computational power available.

On the counterpart CFD, needs high computational power to deal with larger mesh size, due to the mesh density refinement, which is necessary to accurately predict flow with high adverse pressure gradient and strong vortices. This paper experience shows that once the gradients increase, CFD needs a more refined mesh to handle gradients correctly and also demonstrates the need for adequate computational hardware. Such an investment will reduce the time of the design phase, increases accuracy and helps eliminate the uncertainties of an experimental campaign, which remains necessary not only to assess the advantages of a proposed configuration, but also to go beyond the limit of testing due to facility limitations (e.g. blockage).

Consequently, this approach can allow the use of a relatively small wind tunnel for testing larger specimens, but still ensure the validity of the results: it is no longer the performance that needs to be examined and validated, but the validity of the design methodology. Wind tunnels can thus be used to assess the adequacy of design tools and methodology. In this way, thanks to the massive and rapid use of CFD simulations, a larger amount of validated data can become available. Consequently, a larger number of configurations can be explored, making the path to innovation, to the unveiling of ground-breaking solutions and technologies, easier and faster to explore.

ACKNOWLEDGMENTS
The project MOTHIF (MOdel Testing of HIgh LiFt system) has received funding from the Clean Sky 2 Joint Undertaking (JU) under grant agreement No. 865267. The JU receives support from the European Union’s Horizon 2020 research and innovation programme and the Clean Sky 2 JU members other than the Union.

DATA AVAILABILITY STATEMENT
Data that support the findings of this study are available from the corresponding authors upon reasonable request.

DISCLAIMER
The present paper reflects only the author’s view and JU is not responsible for any use that may be made of the information it contains.

REFERENCES
[1] O. Peroomian, S. Chackravarthy, U. C. Goldberg: ‘A Grid Transparent Methodology for CFD’ AIAA 97-0724