Industrial Applications of Computational Fluid Dynamics to Helicopter Development

L. Sudre, S. Borie, P. Alliot, S. Finck

Eurocopter
Aéroport International Marseille Provence
13725 Marignane Cedex, France
e-mail: laurent.sudre@eurocopter.com

Abstract

This paper presents an overview of the extensive use of Computational Fluid Dynamics codes during the development of the EC175. These are the results of efforts made by Eurocopter during the last 20 years in terms of CFD tools development.
Many aspects of helicopter aerodynamics will be presented here.
Internal aerodynamics will be assessed with a particular attention to the engine integration; external aerodynamics will be examined omitting the rotor aerodynamics; finally the CFD capacity to be used in an industrialization process will be considered.

1 Introduction:

By the end of the 90s, Computational Fluid Dynamics (CFD) was developed and validated in Eurocopter thanks to research projects such as CHANCE [2] and SHANEL [3]. Today it is possible to use the full potential of such tools to reduce risks, costs and helicopter development time.
For many years, CFD was too costly to be integrated into a time limited development process. Recent enhancements in CFD methods and in computing capabilities let Eurocopter be able to launch a wide range of CFD applications.
In this paper, we will see the most remarkable applications of CFD applied to the last helicopter developed by Eurocopter: The EC175

The EC175 is a medium range capacity helicopter developed by Eurocopter and HAIG (Chinese aircraft manufacturers). Eurocopter is in charge of General Design, so particularly of aerodynamics studies.

Several aspects of aerodynamics will be presented in the following paper:

- Internal aerodynamics applications
- External aerodynamics applications
- Aero thermal applications

2 Acronyms and notations:

CFD	Computational Fluid Dynamics.
HOST	Helicopter Overall Simulation Tool [5]
HPC	High Performance Computing.
CHANCE	Complete Helicopter Advance Computational Environment [2].
SHANEL	Simulation of Helicopter Aerodynamics, Noise and Elasticity [3].
W*	Reduced air flow: $W^* = \frac{Q_{ejector}}{Q_{exhaust}}$.
T*	Reduced temperature: $T^* = \frac{T_{tot-ejector}}{T_{tot-exhaust}}$.
ΔP	Delta pressure: $\Delta P = P_{tot-ejector} - P_{tot-\infty}$.
q_n	Ejector dynamic pressure.

3 Internal aerodynamics:

Two domains of internal aerodynamics were studied:

1. Engine installation concerning air intake and exhaust nozzle.
2. Upper deck cooling system.

Numerical aerodynamics studies concerning internal air flow were conducted using the ANSYS Fluent code. The size of the unstructured meshes used in the computations was of the order of 5 to 15 millions cells.

3.1 CFD as a pre-design tool for engine integration:

During the predevelopment phase of the project, a trade off study was launched in order to find the best compromise between a short air intake and a long exhaust versus a long air intake and a short exhaust.
This study was performed during the early development phase in collaboration with Pratt & Whitney Canada team.

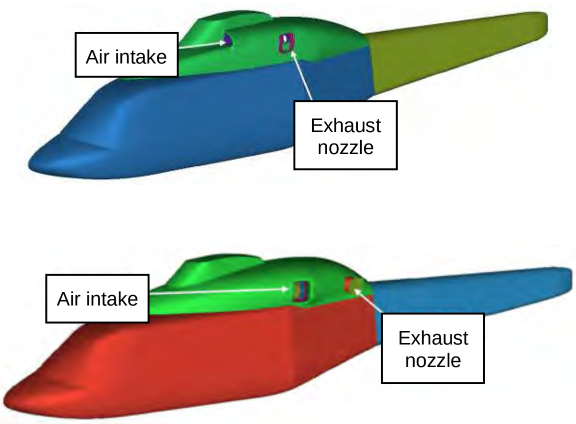


Figure 1: Long intake / short exhaust (up) vs. short intake / long exhaust (down)

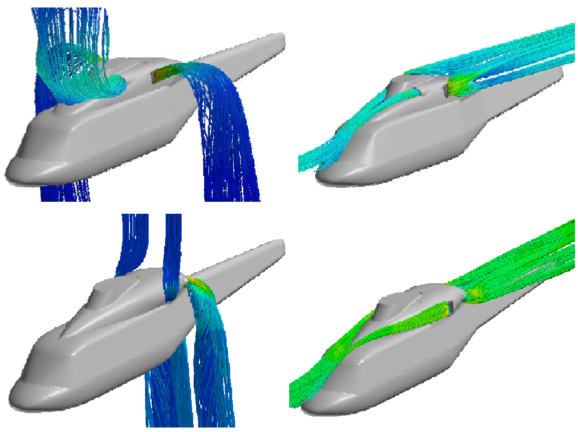


Figure 2: Hover vs. forward flight configuration

The results (regarding aerodynamics, cost and reliability) showed that the best compromise was the configuration which minimizes the exhaust nozzle length.

The EC175 is using a PT6 engine from Pratt & Whitney Canada. This engine mounted on the EC175 has the particularity to be inverted (compressor facing rearwards).
Due to the specific characteristics of the chosen configuration, more attention had to be paid to the intake and exhaust geometries.

3.2 CFD for engine air intake optimization:

As seen before, the long intake / short nozzle configuration was selected. From this first design, several optimized solutions have been derived.

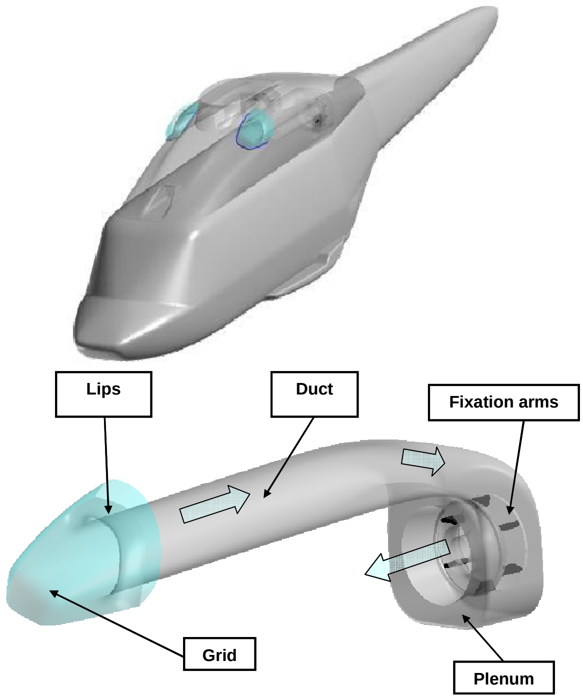


Figure 3: Engine air intake configuration

The initial study has started with an isolated air intake. Quickly it has been shown that the fuselage effect on the flow distortion entering in the duct was very significant. The global optimization has then been conducted on a complete configuration.
Even if no air flow specification had to be fulfilled, the first results showed a strong unacceptable swirl at the compressor plane.
It has been decided to modify the duct's shape and to add a guide vane in order to reduce the swirl to the minimum, and to optimize the pressure losses.

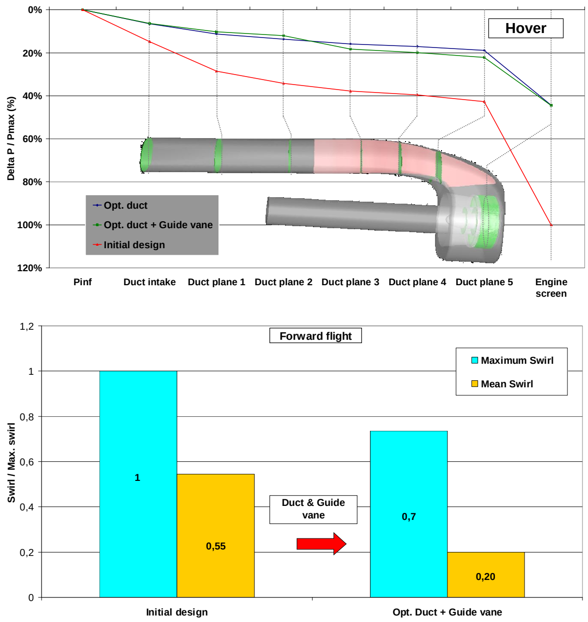


Figure 4: Pressure loss and swirl reduction in several flight conditions

The whole optimization process lasted only three months, letting us update in time the geometry used for wind tunnel tests.
In order to validate the CFD computation, a specific simplified configuration (not taking into account the junction in the duct) was tested.

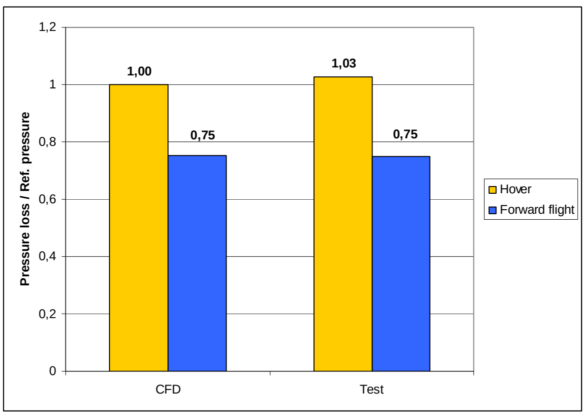


Figure 5: Comparison between CFD results and wind tunnel measurements

3.3 CFD oriented nozzle design

The engine nozzle of the EC175 is used to create cooling air flow thanks to the suction effect of the engine exhaust gazes.
The design of this nozzle was made by the extensive use of CFD.
A virtual test bench was set up in order to evaluate all the configurations in the same conditions.

For each configuration, a characteristic curve was computed aiming to classify the suction performance.

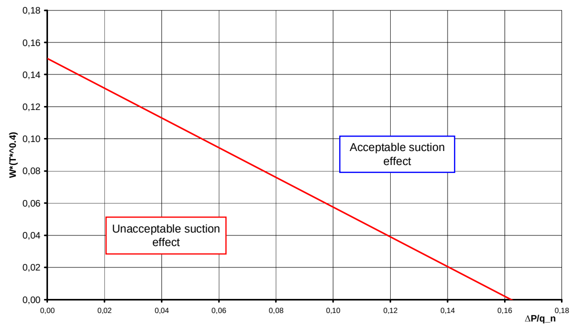
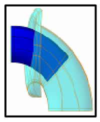
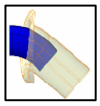


Figure 6: Characteristic curve of suction effect

Figure 6 presents the limit taken by Eurocopter as an acceptable suction effect; this limit is plotted on the graph giving the reduced dilution ratio versus the reduced pressure loss.
First of all, the general design of the exhaust system was computed in order to select the best configuration:

1. Curved secondary nozzle 
2. Straight secondary nozzle 

Results showed clearly that a straight secondary nozzle was preferable. Then a study was performed in order to orient correctly the primary gas flow inside the secondary nozzle.
The 3D virtual test bench was used once again.

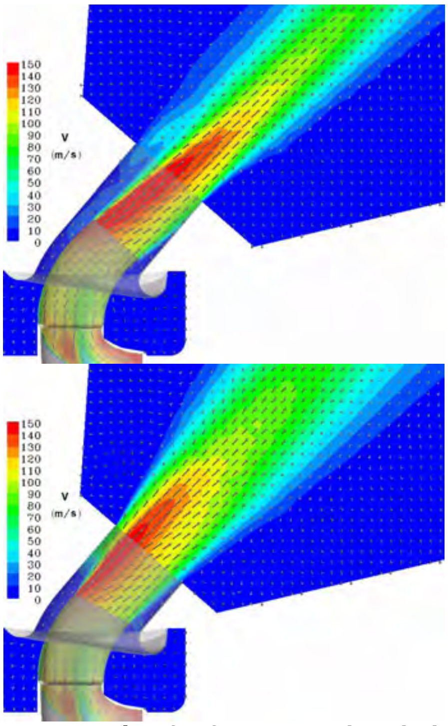


Figure 7: Example of reference and optimized design of engine exhaust systems

Thanks to CFD it was also possible to evaluate the ratio between pumping efficiency and structure weight of the exhaust nozzle and as a matter of fact to optimize this ratio in terms of weight.

3.4 CFD to optimize the upper deck cooling system:

In the front part of the cowlings of the EC175 is situated a ventilation pack constituted of 3 heat exchangers, a fan and a duct.

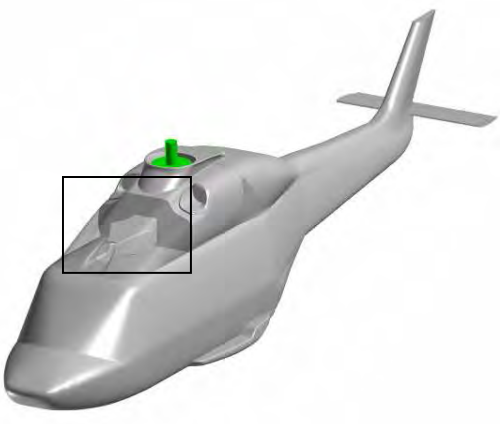


Figure 8: Location of the upper-deck ventilation pack

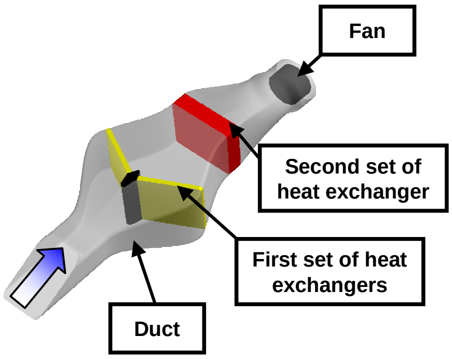


Figure 9: Ventilation pack

Strong requirements in terms of mass flow and pressure losses led to the optimization of this pack very early during the development. As tests bench were yet not available, CFD was used to perform this analysis. The main difficulty was to obtain (like for any helicopter air intake) the efficiency in hover and in forward flight conditions.

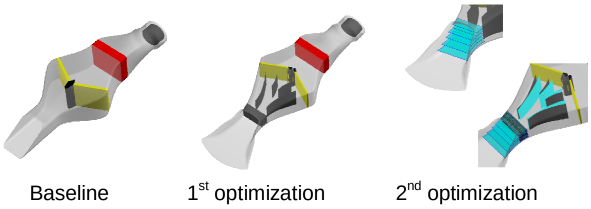


Figure 10: Upper deck ventilation pack evolution

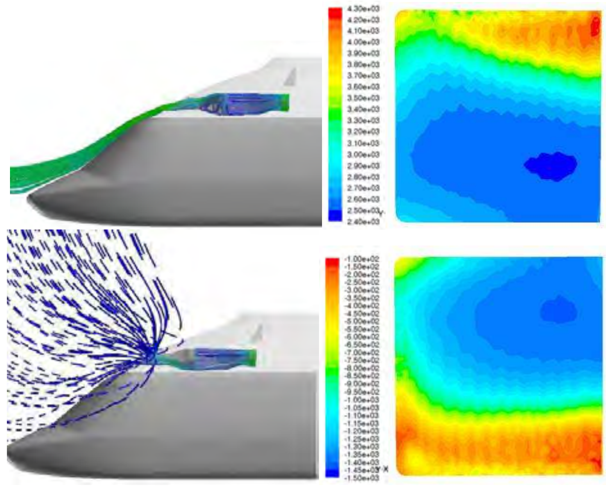


Figure 11: Example of Total pressure distribution on the 1st set of exchangers before optimization.

Figure 11 & Figure 12 show the total pressure distribution on the first set of exchangers in hover and forward flight conditions before and after optimization.
The total process led to the reduction of total pressure losses by 25%.
Thanks to CFD, it was possible to test more than 10 configurations before the manufacturing process start.

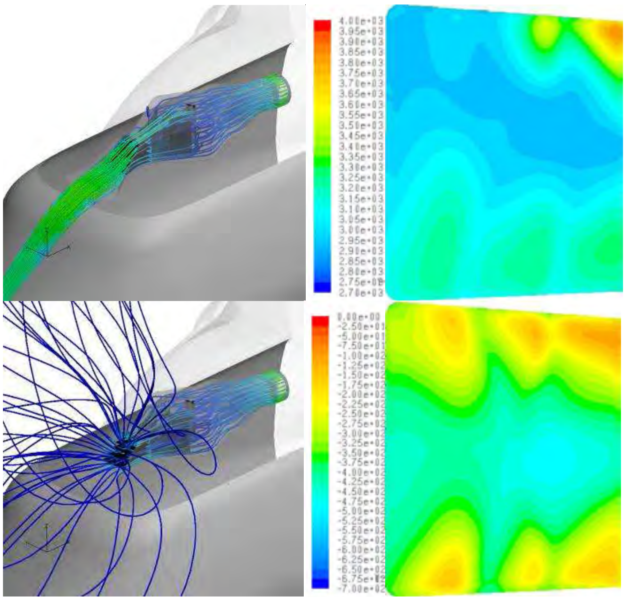


Figure 12: Example of Total pressure distribution on the 1st set of exchangers after optimization.

4 External aerodynamics:

Several aspects of external aerodynamics were examined during the development of EC175.
CFD has been used to:

1. Orient the design in a very early stage;
2. Help Eurocopter to adapt its test bench facilities;
3. Optimize the external shapes;
4. Contribute to the helicopter fine tuning.

Numerical aerodynamics studies concerning external air flow were conducted with the ONERA elsA code. The size of the structured meshes used in the computations was of the order of 10 to 30 millions cells.

4.1 CFD as a guideline to design:

The manufacturing process of the junction between fuselage and tail boom is closely linked to its shape.
In order to evaluate the impact of each proposed geometry, CFD was used to estimate the drag penalty.
A comparison was made with the NH90.

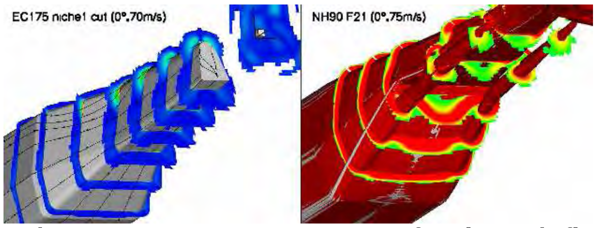


Figure 13: EC175 vs. NH90 rear fuselage air flow comparison.

This comparison shows clearly that the continuity break of the EC175 has no major influence compared to the NH90's rear fuselage and tail boom junction on the air flow.

4.2 CFD as a test bench validation:

During the development of a new helicopter like the EC175, bench tests are mandatory.
In order to anticipate the eventual need of a bench modification, a CFD study concerning the impact of ground proximity on the rear rotor performance has been performed.
Three different heights were simulated. As the environment was not symmetrical, the computations had to be unsteady. In order to modify easily the collective pitch of the blades, chimera technique was used.

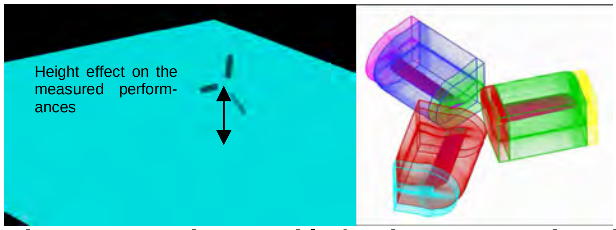


Figure 14: Mesh assembly for the rear rotor bench

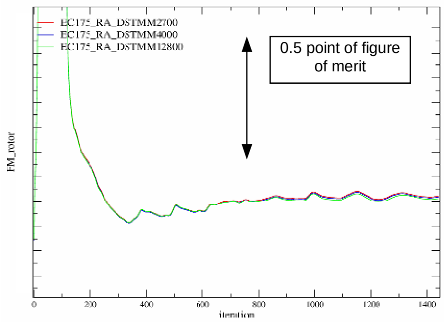


Figure 15: Effect of the height on the rotor's figure of merit

Figure 15 shows the convergence of figure of merit for 3 different heights (2.7 [reference], 4 and 12.8 m).
No significant effect has been detected; no bench modification has been done.

An other application of CFD was to set up the sting interference correction during the wind tunnel campaign.
During the EC175 development, computations of fuselage aerodynamic coefficients were provided before the first wind tunnel test campaign.

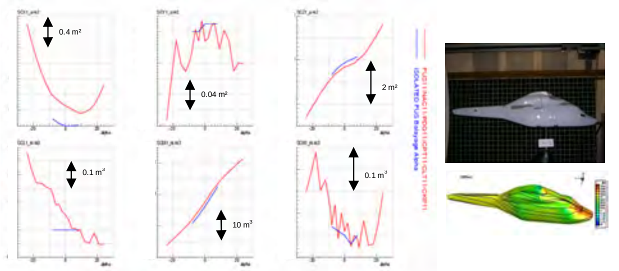


Figure 16: Example of incidence angle sweep from WT and CFD

During this campaign, a technical problem did not let us evaluate the effect of the sting on the aerodynamic coefficients. Thanks to the CFD results their good agreement with wind tunnel measurements, it has been decided to stop the wind tunnel campaign.

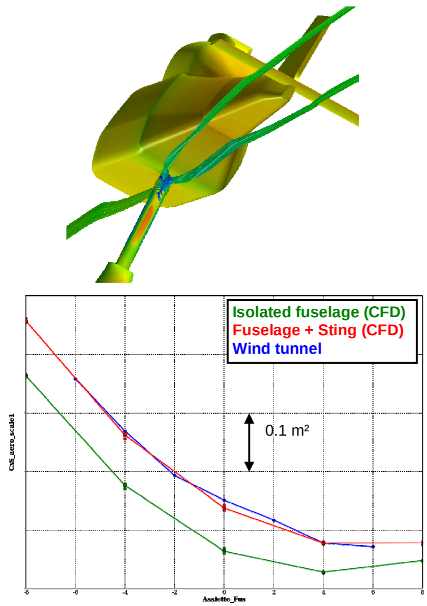


Figure 17: CFD vs. WT - Drag versus incidence angle

4.3 CFD to optimize the shapes:

Drag is a key element to reduce environmental impact of helicopters.
In order to minimize drag, we have to know which is the highest contributor. Thanks to CFD, it is possible to quantify the contribution of each element (or sub-element) of the external shape.
A study concerning the rotor hub drag has been performed.

This work was already presented during the last Rotorcraft Forum [4].
The use of chimera technique permitted to mesh a very complex geometry such as a rotor hub.
A dedicated wind tunnel campaign has been performed in order to build a validation data base.
The *elsA* [1] code have been validated with fixed rotor head, and then with a rotating one.

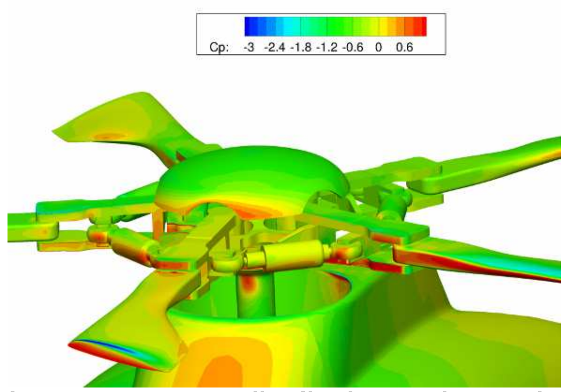


Figure 18: Pressure distribution on the rotating rotor head

4.4 CFD to tune the helicopter

We have seen that CFD was extensively used during the helicopter definition, but the role of CFD cannot halt at this stage of the development.
Long before the first flight, CFD has been used to correct HOST [5] trim analysis in order to get the best possible representation.
The processes described in [4] have been applied to the EC175.

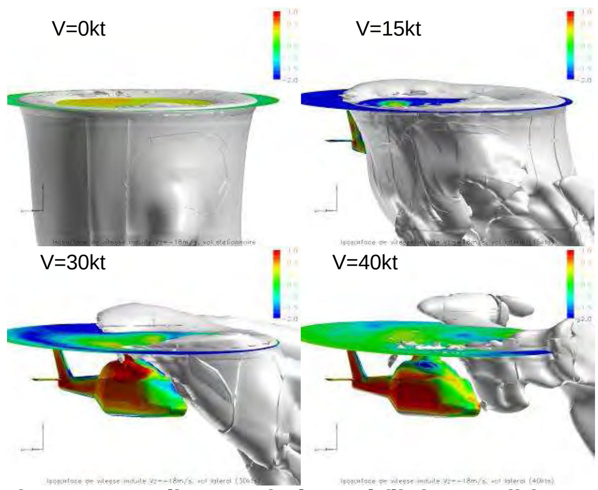


Figure 19: Helicopter in lateral flight conditions - Iso-surface of induced velocity (Vz=-18m/s)

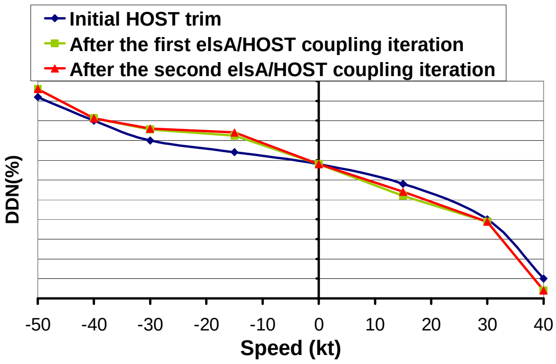


Figure 20 : Position of the control pedal of the tail rotor versus speed

5 Aero-thermal applications:

Interactions between engine intake and exhaust flow and helicopter structure have been evaluated very soon during the development process in order to reduce risks and to limit the number of flight tests.

For this activity, couplings between aerodynamics codes and thermal codes have been used.

In this paper, we will only address the aerodynamics part of the study.

The Fluent code was selected.

Two aspects were particularly examined:

5.1 Re ingestion by CFD

Re–ingestion is a particular phenomenon where exhaust gases ejected from the nozzle are re-ingested by one of the helicopter's engine. This can lead to the engine surge or to a high power loss.

As a helicopter manufacturer, Eurocopter has to certify that the flight cases leading to re-ingestion are not in the helicopter operational domain.

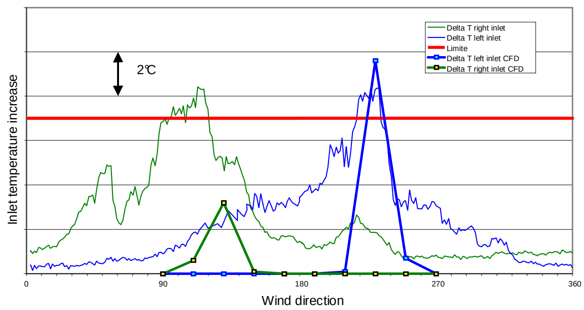


Figure 21: Comparison between wind tunnel and CFD for re-ingestion

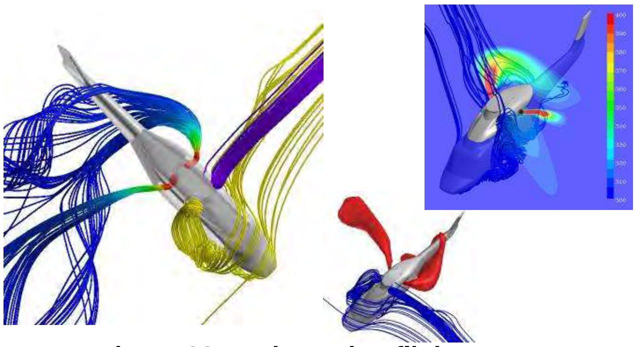


Figure 22: Re-ingestion flight cases

The comparison between wind tunnel measurements and CFD results were very encouraging and showed a good agreement of the position in azimuth of the re-ingestion zone; however the value of temperature was not always at the same level.

Improvement in terms of simulation has to be performed: finer mesh, more precise turbulence modelling.

5.2 CFD as a tool for engine bay cooling definition

Engine cooling system can be very critical for a new helicopter. A poorly designed cooling system can lead to a grounded prototype associated to costs increase and planning shift.

CFD has been used to create a data base of cowling configurations based on the exhaust system defined previously. The best computed solution had been selected as the reference definition for bench tests.

During these tests, CFD results were used to select configuration and optimize the engine cooling.

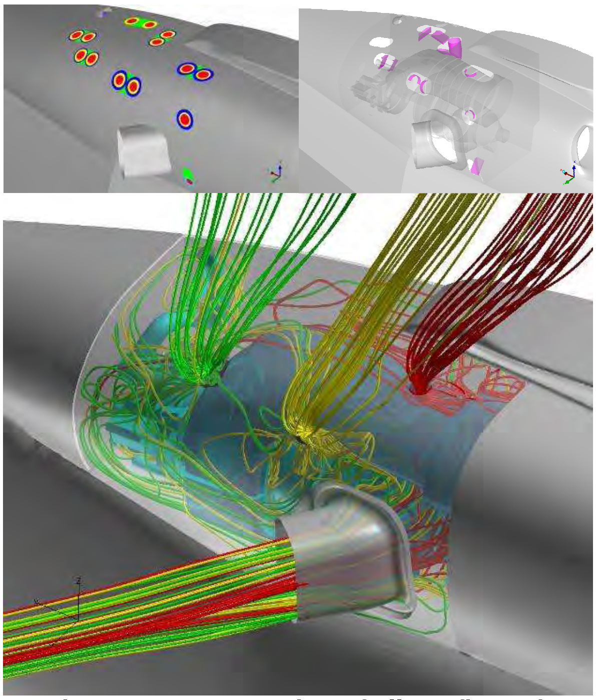


Figure 23: Presentation of all configurations virtually tested (top) - Example of one configuration (bottom)

6 Conclusions:

The applications presented here are examples of the most remarkable usages of CFD in the time limited development of the EC175.

For the first time in Eurocopter, CFD have been applied during the very early stage of the development.

It has clearly shown all the advantages of such a predictive tool.

CFD has been used to design the whole engine integration (e.g. air intake, nozzle and engine cooling), it has also been used to optimize the external shape of the EC175, and finally it has been used in an industrialization process (guideline to design, and test bench validation).

The longest of these studies lasted six months and the mean response time was about one month, which is completely acceptable in a helicopter development timeline.

With the development of CFD methods and increase in HPC power, unsteady applications can now be taken into account (e.g. rotor in forward flight, complete helicopter coupling with aeromechanical tools and structural computation methods [6], etc.) in the industrial development process leading to an even larger CFD application field.

7 References:

[1] L. Cambier, J.P. Veuillot: "*Status of the elsA CFD software for flow simulation and multidisciplinary applications"*, 48th AIAA Aerospace Science Meeting and Exhibit, Reno, USA, 7-10 January 2008.

[2] M. Costes, K. Pahlke, A. D'Alascio, C. Castellin, A. Altmikus: *"Overview of results obtained during the 6-year French-German CHANCE project"*, AHS 61st Annual Forum, Grapevine, TX, June 2005.

[3] M. Costes, J. Raddatz,S. Borie, A. D'Alascio, M. Embacher: *"Advanced Rotorcraft Aeromechanics Studies in the French-German SHANEL Project"*, Proceedings of the 35th European Rotorcraft Forum, Hamburg, Germany, September 2009.

[4] S. Borie, J. Mosca, L. Sudre, C. Benoit, S. Péron: *"Influence of rotor wakes on helicopter aerodynamic behaviour",* 35th European Rotorcraft Forum, Hamburg, Germany, September 2009.

[5] B. Benoit, A.M. Dequin, K. Kampa, W.v. Grünhagen, P.M. Basset, B. Gimonet: *"HOST: A General Helicopter Simulation Tool for Germany and France"*, American Helicopter Society,56th Annual Forum, Virginia Beach, Virginia,May 2000.

[6] M. Dietz, D. Schimke, M. Embacher: *"Advanced industrial application of CFD for helicopter development"*, 36th European Rotorcraft Forum, Paris, France September 2010.