Industrial application of CFD to support certification and qualifications processes at Eurocopter

R. Krauss, F. Dax, A. D'Alascio, D. Schimke
Aerodynamics
Eurocopter Deutschland GmbH
81663 Munich, Germany
e-mail: robert.krauss@eurocopter.com

The paper presents a summary of CFD (Computational Fluid Dynamics) applications at Eurocopter to support the qualification and certification of its helicopters. As CFD methods advance due to higher computer power, more accurate physical and numerical modelling and higher mesh densities, simulation accuracy increases as well as the trust in the results of CFD simulations by aviation authorities. Consequently new opportunities arise to apply CFD. One of such opportunities is the support of qualification and certification processes, wherein the functionality of the helicopter or parts has to be proved towards the customer/client (qualification) or aviation authorities such as EASA or/ and FAA (certification). Based on examples the opportunities, difficulties and particulars to be considered are explained.

INTRODUCTION

During the last decade Computational Fluid Dynamics experienced an important progress driven by research centres and software development companies and a consequent broadening into industrial processes. This is due to several reasons:

- improved pre-processing, i.e. direct import of CAD surfaces and faster and easier mesh generation, and post processing, e.g. capability of monitoring the solution during execution, plotting quantities in the field and on the surface;
- the increasing complexity and accuracy of the physical modelling;
- the higher accuracy of the numerical schemes;
- higher confidence in the results thanks to method validation about cases of industrial relevance;
- availability of tools running efficiently on massive parallel computing such as Linux clusters.

This promoted the integration of CFD into the industrial design process at Eurocopter, in a concurrent engineering fashion with other disciplines, at an early design phase, during detailed design - thanks to the introduction of CATIA v.5 inside the aerodynamic department - and even after production to support the certification and qualification processes. The use of CFD in this late stage is meant to reduce or even eliminate flight or bench tests, which are normally required by the certification authorities or by the customer, respectively to certify or qualify a new machine or one of its subsystems. Depending on the complexity of the (flight or bench) tests compared to the CFD simulation effort, costs can be saved. There have been also cases, in which CFD simulation was used to specify certification or qualification flight conditions, which might have been dangerous or polluting the environment [2].

Not only advantages in using CFD for certification or qualification shall be considered but also disadvantages. One of such is the possibility that the CFD simulation is not accepted by the customer or authority and flight test have to be done anyways. This will produce additional costs and an increased timeframe to complete the certification process. Therefore it is important to work with the customer or the authorities from the beginning and clarify whether a CFD simulation might be taken into account. At this early stage validated CFD should be able to prove good correlations between simulation results and flight or bench test measurements.

In several different projects at Eurocopter CFD simulations were successfully applied for certification and qualification processes. The following chapters give some insight into these CFD applications.

CERTIFICATION OF A HELICOPTER'S FIRE EXTINGUISHER SYSTEM

The power plant compartment of a helicopter has a fire extinguisher system which has to be in compliance with the FAR (FAA) and / or CS (EASA) regulations. For a first version of the helicopter it was proven in flight tests that the fire extinguishing system complies with these regulations. The latest development is based on its predecessor. Several small changes are made to the original power plant compartment, which might influence the performance of the fire extinguisher system. Since the changes are relatively small, it was decided that their possible influence could be analysed via a validated CFD model. The regulations require that a certain fire extinguisher agent concentration is reached at all points in the volume of the engine bay simultaneously. If there is a change to the engine bay which might have an influence on the effectiveness of the fire extinguishing system, the certification has to be done again, resulting in additional flight tests. Those tests are very expensive, time consuming and are also harmful to the environment since Halon 1301 (fire extinguisher agent) dissolves the ozone layer. An alternative method to provide a means of compliance is therefore very useful. Computational Fluid Dynamics (CFD) can potentially be used to verify the function of the fire extinguishing system, provided that the CFD prediction is accurate enough.

Therefore, a CFD model of the engine bay's fire extinguishing system was developed. The predictions of this CFD model were compared to flight test data to validate the model. It could be shown that the CFD model accurately predicts the time evolution of the Halon concentration after it is discharged into the engine bay. Consequently the model can be used in the future for comparative analyses, in which effects to the fire extinguishing system, due to small changes in the model boundary and initial conditions; such as changes to the engine bay geometry or to the fire extinguisher system itself or to the initial state conditions of the fire extinguisher agent in the bottle; can be investigated. The model is not accurate enough to precisely predict absolute values for the concentration of the fire extinguishing agent: the time evolution of the predicted mass concentration does not match exactly the measured values as shown in Figure 1. Nevertheless the time interval in which the mass concentration exceeds the 22% value, dictated by the certification regulations is the same in CFD and experimental measurements, which is crucial for certification purposes.

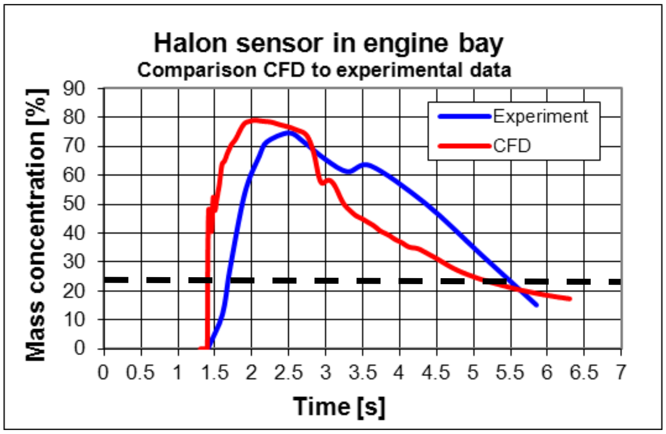

Figure 1: Comparison of the time history of the Halon mass concentration between flight test measurements and CFD predictions about the original version of the helicopter; the black dashed line indicates the minimum concentration, according to certification regulations to extinguish a fire

The CFD model was used [1] to compare the function of the fire extinguishing system from the original version of the helicopter to its upgraded version (see Figure 2). For this comparison the fire extinguisher agent (Halon 1301) concentration levels of the upgraded helicopter were calculated at several points in the engine bay and compared with the same concentration levels from the original power plant compartment. Changes to the expansion and distribution of the Halon 1301 in the engine bay could be analysed.

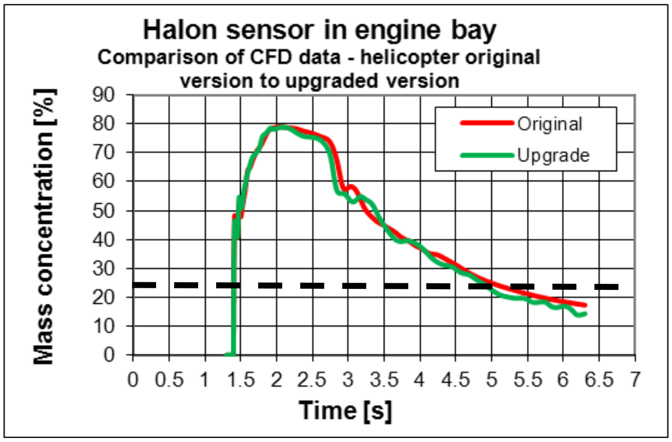

Figure 2: Comparison of the (by CFD) predicted Halon concentration levels of the original version of the helicopter and the upgraded version; the black dashed line indicates the minimum concentration to extinguish a fire

It was shown that the Halon 1301 concentration over time at all points in the engine bay and also the period in which all sensors reach the necessary concentration to extinguish a fire remained very similar, which implies that the upgraded helicopter's fire extinguishing system will still comply with the regulations. Accordingly the certification of the helicopter's fire extinguishing system by CFD was accepted by the EASA.

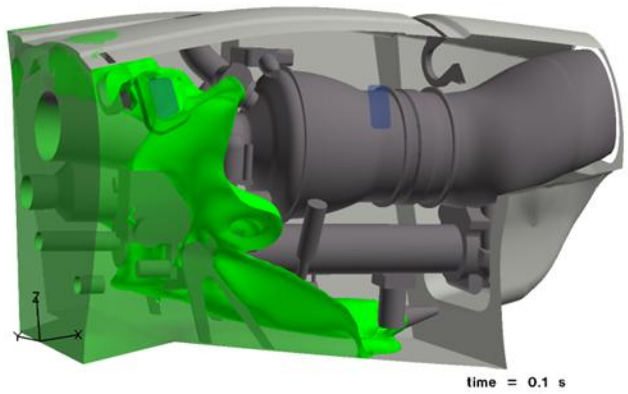

Figure 3: Distribution of Halon 1301 – 0.1sec after discharge

Conclusion

The successful certification of the helicopter's fire extinguishing system could be achieved since the validation standard of the CFD model was judged to be accurate enough for this application by the EASA. The fact, that it was a delta analysis, where only differences were analysed due to small geometric changes also helped for the acceptance of CFD. This application was an ideal example for the use of CFD in certification processes. No costs for flight tests or experiments aroused.

QUALIFICATION OF A NEW FUEL VENTILATION OUTLET

A fuel ventilation system, a sketch of which is depicted in Figure 4, has the function of ventilating the fuel tanks. It means during re-fuelling air is pushed out of the ventilation outlets while the tanks are filled up and during flight air is sucked in through the ventilation outlets while fuel is consumed. If the ventilation outlets are clogged by ice, for instance, the fuel indication level might be overestimated. This is of course not accepted by the customer. To assure that the fuel level is correctly displayed, a ventilation outlet is designed, which assures overpressure during forward flight and a neutral behaviour in hover in the fuel ventilation master lines.

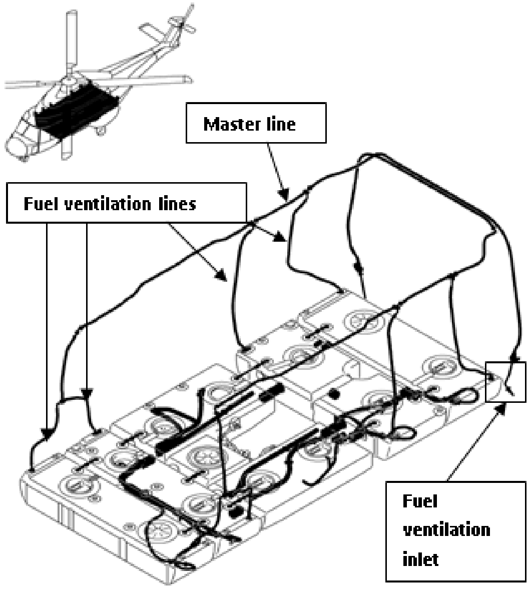

Figure 4: Schematic view of a fuel ventilation system of a helicopter

A CFD investigation was carried out by Eurocopter to improve the icing behaviour of the external ventilation outlets of the fuel tank of a large transport helicopter. The requirement from the program management was to design a retrofit solution. The aim of this analysis was, *first*, to numerically assess the aerodynamic behaviour of the baseline ventilation outlet V0 shown in Figure 5-*left* and Figure 7-*left*, *second*, to suggest modifications to this geometry and prove that their aerodynamic behaviour does not degrade. *Finally*, after having conducted icing wind tunnel tests with the proposed isolated ventilation outlets (not integrated on the helicopter's floor, as shown in Figure 5), which were necessary to choose the best candidate and the most suited installation angle, a CFD analysis was carried out to support flight test and qualification of the selected new geometry.

The vent outlet V1 depicted in Figure 5-*right* and Figure 7-*middle*, which was identified through icing wind tunnel measurements as the best performing candidate in icing conditions, was tested in flight in non-icing conditions, to assess its aerodynamic behaviour. This proved to be as good as the baseline solution and provided validation data for CFD computations. The icing wind tunnel tests were conducted for several flow incidence angles. The slowest ice accretion was registered for an incidence of 0° to the up-stream air flow. A sketch of the ventilation outlet installation scheme in the icing wind tunnel test chamber is given in Figure 6.

Figure 5: Ventilation outlets V0 (left) and V1 (right) mounted in the icing wind tunnel test section

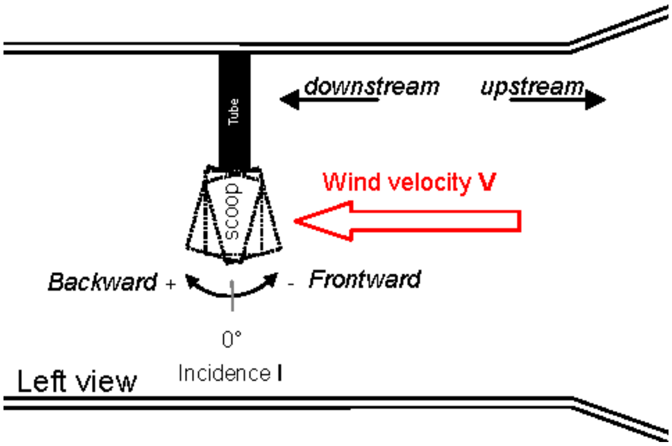

Figure 6: Fuel ventilation outlet (scoop) installation scheme in the icing wind tunnel

Considering that the ventilation outlet V1 installed on the helicopter features an incidence of -15° to the up-stream air flow during flight, the ventilation outlet V1 was redesigned, now featuring a 15° forward rotation with respect to the ventilation pipe: V2 of Figure 7-*right*. An additional CFD analysis was necessary to compare the ventilation outlet V1 tested in flight and the identical one measured in the wind tunnel with a 15° forward rotation V2.

Figure 8 shows the pressure coefficient inside the fuel tanks ventilation master line (depicted in dark blue) just after the ventilation outlet. In the diagrams the geometry of the ventilation outlet V2 has been taken as reference to show exactly where the pressure values are extracted (red spots). The ventilation master line is of course the same for all investigated outlets. The values for the original (V0) and the two modified outlets (V1 and V2) are compared. It is evident that the new 15° forward rotated ventilation outlet (V2) performs slightly better than V0 and V1, in fact it is able to achieve higher static pressure values in the fuel tanks vent master line. The pressure coefficient gives an indication of the "dynamic pressure recovery" capability.

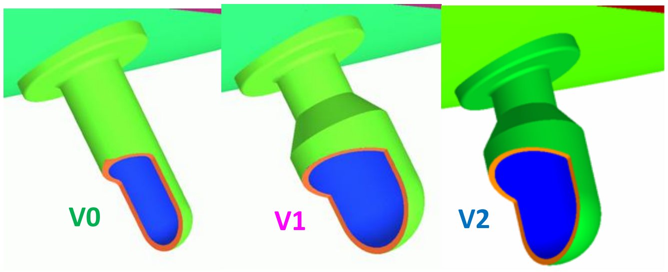

Figure 7 Original ventilation scoop (left), candidate selected for icing wind tunnel tests (middle) and final qualified ventilation scoop (right)

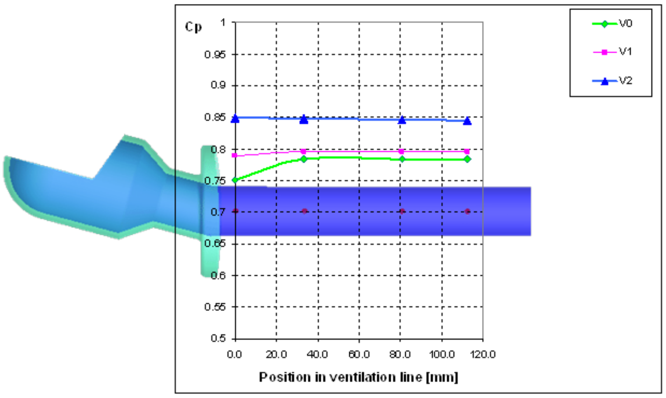

Figure 8: Comparison between vent outlets V0, V1 and V2 in terms of pressure coefficient inside the fuel tanks master vent line.

The results of the current investigation showed that, between the two fuel tank ventilation outlets analysed, the second (V2) featuring a 15° forward rotation – i.e. incidence 0° to the up-stream air flow once it is installed on the helicopter and flown -, performs slightly better than the first straight one (V1) – i.e. incidence -15° to the up-stream air flow -; both of them being somewhat better than the initial vent outlet (V0)

– i.e. incidence -15° to the up-stream air flow installed and in flight as shown in the CFD results of Figure 9.

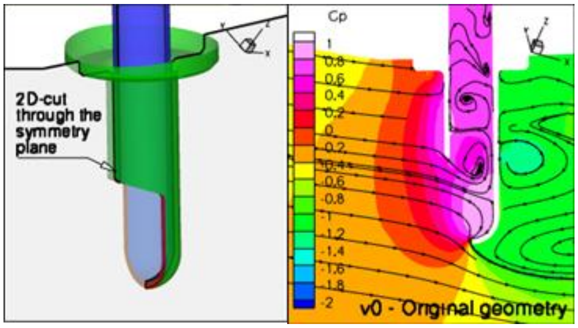

Figure 9: Axial cut through the original ventilation outlet installed on the helicopter floor (left); pressure coefficient and stream velocity plots (right).

Conclusion

The CFD comparison between the ventilation outlet V1 and V2 was accepted by the customer, without having to perform additional flight tests in non-icing conditions with the final selected ventilation outlet geometry.

DAMPING FUEL SLOSHING MOVEMENTS

Refuelling a military helicopter on a ship deck proofed to be problematic at rough seas. The ship movements cause kerosene movements that stop the refuelling process early. The kerosene sloshes to the left and right side of the tank. As a result the fuel gauge temporarily measures fill levels which represent a full tank. This automatically shuts off the pressure refuelling process. Testing in rough sea conditions showed that the helicopter’s tanks could only be partly filled. Consequently the helicopter’s range would be reduced.

A two-dimensional unsteady (time dependent) CFD simulation of the tank was carried out to analyse the fuel motion in the tank. The rolling motion of the vessel was measured and then used for the simulation: +/-10° in an 11sec period. The centre for the rolling motion was assumed to be 5-10m below the landing deck for the helicopter (depending on the vessel). The tank was 50% filled with kerosene. The phenomena of the refuelling test from the ship deck could be reproduced. Figure 10 shows how the kerosene is sloshing from side to side. Since the sensor for the fuel gauge lies on the side of the tank, it can be seen that it measures a fill level that represent a full tank as the kerosene sloshes to the side, were the sensor is placed.

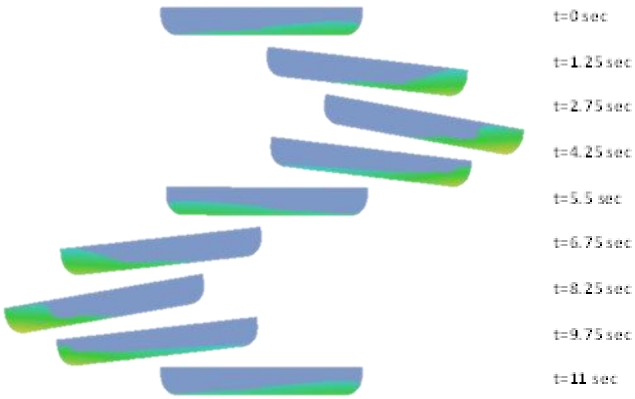

Figure 10: Kerosene sloshing movements in the tank without any damping device

A damping system, that limits the fuel sloshing, proposed by tank specialists of Eurocopter, was analysed by CFD afterwards. The damping device consists of high porous foam that can be inserted into the tank without having to modify it. A 0.4m wide block was suggested to be inserted into the tank. Even though the foam block is relatively large the tank capacity remains very similar as the material is highly porous. In a CFD simulation the damping device was simulated as a porous zone in the CFD solver. Figure 11 demonstrates that the kerosene sloshing movements could be successfully damped. . This was also approved by a rig test, which showed that the damping of the kerosene sloshing movements is sufficient.

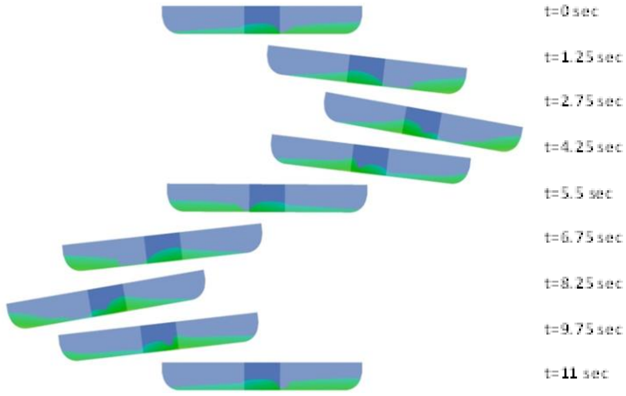

Figure 11: Kerosene sloshing movements in the tank with a porous foam damping device (dark region)

Another result that could be obtained from the simulation was the forces acting on the damping device. They could be used to verify a suitable fixation of the foam inside the fuel tank module.

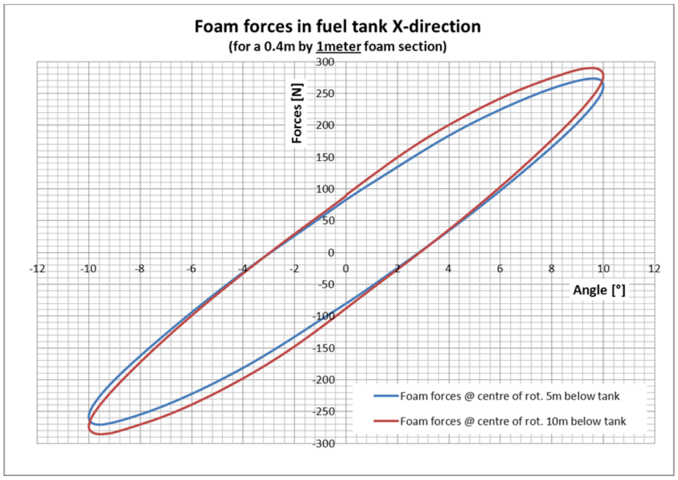

Figure 12: Forces on the foam damping device over ship roll angle.

After the numerical simulation, experiments were carried out to confirm the results obtained with CFD and to further support the qualification process.

Conclusion

In this case the application of CFD helped to significantly reduce the amount of tests and experiments, which otherwise had to be carried out. Time and costs could be saved.

QUALIFICATION OF THE FUEL JETTISONING SYSTEM

Fuel jettisoning is an emergency procedure during which the helicopter's fuel tank is emptied in flight. This operation must be performed in a particular time range. Fuel jettisoning was tested in flight for a large military transport helicopter wherein the time period to dump the fuel was determined. Due to changes to the fuel dump piping system, flight tests or CFD computations were necessary to proof the faultless functionality of the new fuel dump system. Especially the time that is needed to dump the fuel must not increase due to the new piping system. The changes to the pipe needed to be done because it conflicted with the use of a traversing system used to move the helicopter on the ground. CFD proved to deliver accurate results for these kinds of investigation in the past, hence it was chosen as a means of compliance.

A holistic CFD model of the complete fuel dump system was set up, consisting of the helicopter's front and rear tank system as well as the pipes. The computational domain ended out the pipe outlet. The tank and original pipe geometry can be seen in the figure below.

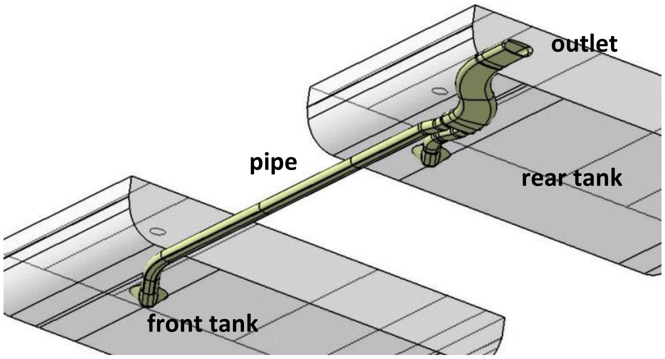

Figure 13: Original fuel dump piping system

A first CFD investigation aimed at the validation of the CFD model with flight test data from the original fuel dump system. The simulation was set up with FLUENT V13 as a steady state simulation. It simulated the kerosene flow driven by the gravity out of a fully filled tank. A comparison of the results (outlet mass flow rate) with the flight test data showed that the predicted value lies within the tolerance of the flight test data. Furthermore, an interpolation of the steady state results from the fully filled tank to other tank fill levels also matched the flight test data within its tolerance. For the interpolation to other fill levels the Torricelli equation was used in combination with a loss factor derived from CFD. The Torricelli equation calculates the outlet velocity of a liquid out of a tank, based on the height of the fluid in the reservoir and the gravity force.

$$v_{outlet}(t) = \sqrt{2gh(t)} \bullet c_{loss}$$

Since the results of the CFD analysis and the interpolation to other fill levels matched the flight test results within its tolerance, the CFD model was considered to be validated.

Following the validation of the CFD model another simulation with identical settings but with the altered pipe geometry was carried out. The new pipe geometry can be seen in the figure below. As in the first investigation, the CFD results were interpolated for other fill levels.

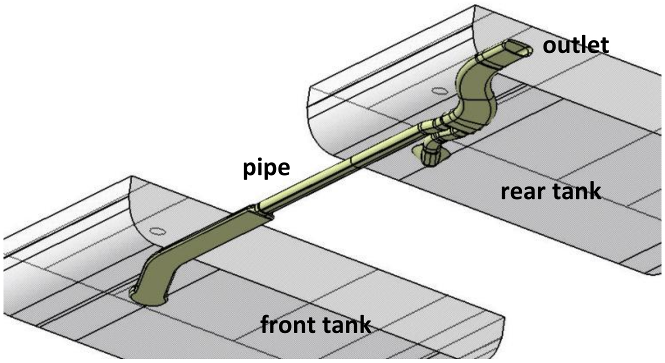

Figure 14: Modified fuel dump piping system

The results from this analysis showed that the fuel jettisoning process will not take longer with the new changed piping system. It could be shown that the rear tank (not influenced by the new pipe) is always the one which is emptied the latest and is therefore the main factor that determines the time needed to empty the tanks.

It was proven with a validated model that the fuel dump process will not take longer with the new pipe. The qualification of the changed piping system by CFD was accepted by the customer.

Conclusion

The changed piping system could be successfully qualified by CFD. Due to the good validation standard of the CFD model the customer had no doubts that the piping system will meet his requirements. Hence there was no need for expensive rig or flight tests. Time and costs could be saved.

GENERAL CONCLUSIONS

The Paper gave various examples, where Eurocopter was able to successfully apply Computational Fluid Dynamics for certification and qualification purposes. Wherever it is dealt with modifications of an existing already certified or qualified system; i.e. fire extinguishing system, fuel dump, fuel ventilation ports, tank system; numerical analysis is accepted by the EASA or by the customer as proof of compliance, granted that the numerical method be validated against bench or flight test measurements. It is mandatory, though, that the validation is consistent, which means it is made against a predecessor system for which reliable measurement data are available from previous certification and qualification activities.
The continuous applications of CFD simulations already in the design process of a helicopter subsystem provides with further potentials:

- lower risk of non-compliance in the certification/qualification process;
- ability to suggest fast design alternatives in case of “numerical non-compliance”;
- higher confidence in the application of CFD tools in industry, gained through new validation cases available (numerical results produced during the design phase are eventually compared against flight test data, once the first prototype flies) ease the acceptance process by the certification authorities and by the customer.

REFERENCES

[1] Berthe A., D’Alascio A., “CFD simulation of fire extinguishing system. Two gas mixture inside the engine bay.”, Proceedings of the 29th European Rotorcraft Forum, September, 2003, Friedrichshafen, Germany

[2] D’Alascio A., “Simulation of fuel jettisoning. Analysis of a complex flow field during flight test using CFD, Proceedings of the 30th European Rotorcraft Forum, September, 2004, Marseille, France