# CFD SIMULATION OF FIRE EXTINGUISHING SYSTEM: TWO GASES MIXTURE INSIDE THE ENGINE BAY

A. Berthe and A. D'Alascio
EUROCOPTER DEUTSCHLAND GmbH, Munich, Germany

**The paper presents a CFD simulation of the fire extinguishing system of the BK117 helicopter. An extinguisher gas is instantaneously released into the helicopter engine compartment if fire sets in. Within 5 seconds the extinguishing gas is burst into the engine bay from two pressurised bottles, mixes with the air inside the compartment and should extinguish the fire.**

**Test measurements have been carried out in the BK117 engine bay and the concentration of the fire extinguishing gas has been recorded at several test points. The certification regulation requests that a certain gas concentration is reached for a minimum time so that the fire would be extinguished. The aim of this study is to reproduce experiments in order to support the design department if the need for modification arises. The CFD calculations should be able to predict the influences of different parameters, like cowling geometry or mass air flow, on the gas mixing process.**

**The engine bay surface geometry has been extracted from a set of existing CATIA models of the engine, engine fairing and fire walls. The volume mesh describing the computational domain of the flow field has been generated by the ICEM Tetra software [1], and the numerical predictions have been calculated with the commercial code FLUENT [2].**

**The test results consist of measurements of the mass concentration of the fire extinguishing agent with the time, recorded in different locations of the engine bay once the agent discharges into the bay. The process is treated as unsteady. Here the first CFD results are presented. The comparison with test results shows, that the gas concentration as a function of the time can be simulated, however the absolute values are too low. A more detailled analysis of the phenomena is needed, to better define the boundary conditions as this process is very time dependent.**

**The CFD results show a positive result in terms of complete simulation of such a process and encourage a more detailed modelisation.**

## Nomenclature

| | |
|---|---|
| $A$ | Inlet section surface area [m$^2$] |
| CFD | Computational Fluid Dynamics |
| FEA | Fire Extinguishing Agent |
| $\dot{m}$ | Mass flow [kg/s] |
| Ma | Mach number |
| $p$ | Static pressure [Hpa] |
| $\rho$ | Density [kg/m$^3$] |
| v | Velocity magnitude [m/s] |

## Introduction

The need to increase competitiveness and to reduce development and certification time has driven the helicopter industry to introduce CFD tools for analysing the flow field not only on external components but also for internal flow and mixing processes. Tests are always very expensive, time consuming and give no insight in the 3D flow process, which is decisive for the design optimisation. The availability of powerful computers has given scientists the possibility to develop numerical algorithms to solve the 3D steady and unsteady Navier-Stokes equations coupled with reacting and non-reacting species equations. The industrial community is now able to simulate mixing, reacting and non reacting, single and multiphase flows. The helicopter industry is therefore increasingly using CFD methods by incorporating them in its design environment [3] in order to reduce the number of experiments with a greater number of configurations being explored numerically. On one hand this speeds up the optimisation process and on the other hand it reduces its costs.

## Fire extinguishing system description

The fire extinguishing emergency system is activated when one of the engines is set on fire. In such a case the pilot shall reach the OEI (One engine Inoperative) flight condition, close the fuel valve, and cut off the engine. As soon as the engine rotation speed is reduced to 50%, the fire extinguishing agent is released from the pressure bottle through a bursting valve and is discharged inside the engine bay. The certification requests the presence of a volume concentration through out the engine bay of 6% for a minimum time of 0.5s.

For the original configuration the gas mixing process was tested and the concentration is measured in several samples distributed into the engine bay.

The complete system consists of 2 bottles of fire extinguishing agent, each of them can be discharged in each of the two engines, this yields to 4 combinations which have to be tested. Due to some non-symmetry, the volume flow of air in both engine bays, which determines the evolution of FEA concentration with the time, is not exactly the same. Measures of the volume flow in the bay have been conducted in different flight configurations in order to determine the worst case. The worst case for the fire extinguishing system is defined as the highest air flow, since in this case the concentration of the FEA is a minimum.

## BK117 engine bay geometry

Figure 1 shows the engine bay, especially the volume comprised between the helicopter engine fairing, the forward and backward firewalls and the engine external surfaces, after the necessary modifications for CFD applications, *i.e.* removing or smoothing of all wiring, screws and small surface irregularities. Figure 2 shows the engine surface before this smoothing operation.
The air enters the engine bay through the inlet scoops (represented in Figure 1 in blue), circulates inside it and is sucked out through the ejector outlet (represented as a dark green taurus). The fire extinguishing agent is released through the small green pipe (fire extinguisher inlet).
The goal of the simulation is to reproduce the experiments, in order to support the design department when modifications are needed. For instance it may be necessary to change the shape of the engine cowling, which has an impact on the mass air flow and consequently on the concentration of the FEA. With the CFD calculations the influence of the design modifications should be identified and should help to reduce the certification tests to a minimum [3].

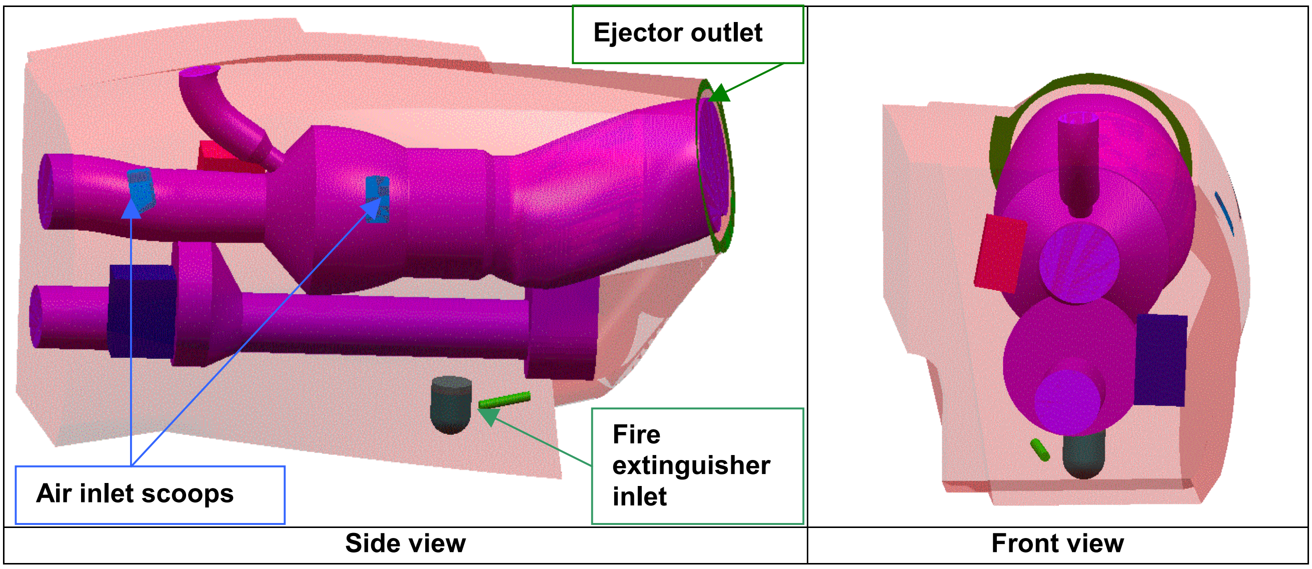



Figure 1: left picture, side view of the engine bay ; right picture, front view of the engine bay after the modifications necessary for CFD application

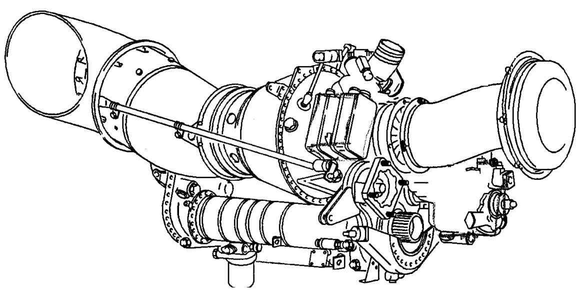
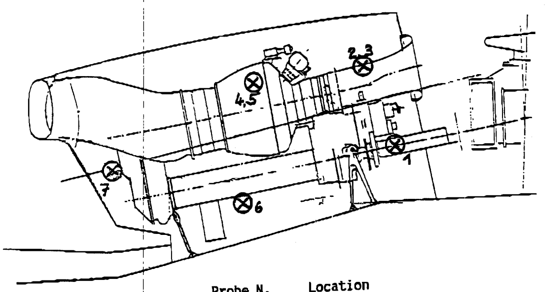


Figure 2: Top picture, original engine geometry

## Description of the tests

The measurements presented here were conducted at CGTM France. The concentration of the FEA has been measured at several locations in the engine bay. The sensor positions are reported in Figure 3.
The tests were performed at different flight conditions: hover flight, descent at 50kts and level flight at 100kts, while both pressurised bottles of FEA were discharged in either the first or the second engine bay.
The test which showed the shortest time of FEA concentration was the level flight at 100kts, when the FEA was released in the right engine bay.

This condition was selected for the CFD calculations. The interval of time during which the concentration of the FEA is above 6% has been measured between the latest rise and the earliest decrease of concentration at two sensors.



| Probe N. | Location |
|---|---|
| 1 | mid of lower forward area |
| 2 | leftsided upper forward area (above fuel control unit) |
| 3 | rightsided upper forward area (same level as probe n. 2) |
| 4 | near the left hand starting fuel injector |
| 5 | near the right hand starting fuel injector |
| 6 | in front of fuel filter |
| 7 | near the N2 magnetic pick up |

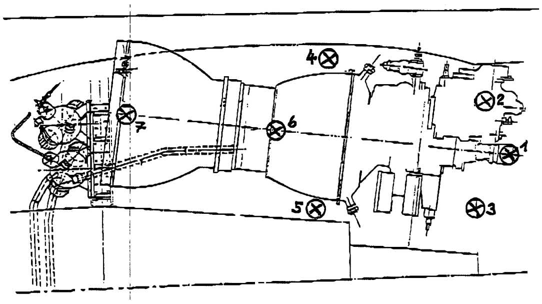


Figure 3: Location of the sensors distributed into the engine bay.

## CFD model description

### Grid generation

Starting from an already existing CATIA model a simplified water-proof surface model (see Figure 1) was designed, which consists only of the engine fairing surface, the fire walls and the engine external surfaces. A volume mesh has been generated by directly importing the simplified CATIA surface model into the ICEMCFD software and by generating a volume grid with the ICEM-Tetra package. The pure unstructured mesh is composed of about 560000 tetrahedrons and 100000 nodes. It has been decided to keep the grid topology as simple as possible by avoiding the generation of a boundary layer prism-mesh. In fact the use of an hybrid mesh with a fine prism model of the boundary layer grid is necessary only when an accurate prediction of the strong gradients occurring in the boundary layer region is foreseen. In this simulation only the fire extinguishing agent concentration inside the volume, and therefore at a sufficient distance from the walls, is of interest. The volume mesh has been finally exported from the ICEM tool into the FLUENT format.

### Initial and boundary conditions

The fluid is defined as a mixture of two non-reacting gases which mix and diffuse inside the volume.
The initial and boundary conditions have been derived from known values at 100kts level flight.
The boundary conditions for the flow solver are defined as follows (see also Table 1):

- on all *solid surfaces* the no-slip condition has been applied;
- at the *air inlet scoops* a constant value for the air mass flow has been chosen, which corresponds to a velocity of 25m/s at the air inlet scoops.
- at the *fire extinguisher inlet* a certain FEA mass flow as a function of time, has been defined. The velocity time law has been derived by resolving the Fanno flow obtained by discharging the pressurised bottle through the small pipe into the engine bay. With such an assumption, the FEA reaches Ma=1 at the outlet or a velocity of v=132m/s. As the bursting valve is closed before being opened (by explosion), a short time is needed until the velocity of 132m/s is established. This time is averaged to 0.005s. This period was simulated with a very small time step and a constant increase gradient of the velocity.
- the outlet is modelled as pressure outlet where both components are going out. In case of a back flow taking place at the outlet section due to local recirculation, only air is assumed to re-enter the engine bay.

| | |
|---|---|
| Air inlet scoops | $v_{Air} = \dot{m}_{Air} / (\rho A)_{Air} \quad [m/s]$ |
| Fire extinguisher inlet | $v_{Halon} = \dot{m}_{Halon} / (\rho A)_{Halon} \quad [m/s]$ |
| Outlet section | $p_{out} = 0 \quad [Hpa]$ |
| Wall | $v_{wall} = 0 \quad [m/s]$ |

Table 1: definition of FLUENT boundary conditions

The operating conditions yield a gauge pressure of 100825 Pa, chosen to simulate the under pressure at the ejector outlet.
The computation was first driven to convergence steadily, only with air, as it is the case in reality before an engine fire. Then the FEA was introduced with a very small velocity and the process changed to unsteady. Then with a small time step the FEA velocity reaches 132m/s after which the time step was slowly increased to 0.01s. At the end of the FEA discharge, at time 1,42s, the velocity was progressively decreased to 0.1 m/s in 0.2s.

The flow has been assumed incompressible. In fact the Mach number inside the engine bay is very small almost everywhere except for the very limited region in the proximity of the FEA inlet section. In the vicinity of the FEA inlet the flow is compressible therefore the results will not be considered as correct in this area (there is no sensor located close to it).
Finally the temperature was set to a constant value and its influence on the FEA mass fraction evolution assumed to be minimal.

**Flow solver parameters**
The incompressible Navier-Stokes solver FLUENT has been run in unsteady mode implementing the SIMPLEC pressure based method. The species transport is modelled by solving conservation equations describing diffusion and convection for each component. A second order scheme in space has been used together with a Multigrid acceleration technique. The standard k-ε 2-equation turbulence model with wall function approximation has been applied.

Flow analysis

A vertical cut at the engine middle and two horizontal cuts: one at the FEA outlet pipe and one through the engine, have been defined through the engine bay and engine to visualise the 3D flow development.
Five timesteps have been chosen for the following pictures:

- very shortly after the FEA bottle opening valve burst
- a few ms later
- at the moment at which the FEA velocity begins to fall
- half a second later
- at the end of the calculation

as illustrated on Figure 4. It can also be observed, that the sensor 1 in the lower forward area is the first one to be affected by the presence of FEA, followed by the sensor 3 in the upper right area of the bay, and finally the sensor 7 at the rear. The same can be remarked for the maximum values of the concentration: where the FEA come first, the concentration will be higher, but the decrease of concentration will be quicker. This is more obvious for the sensor 7.

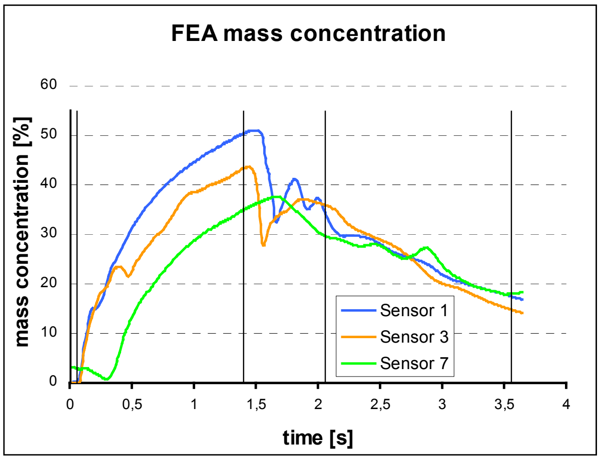


Figure 4: Mass concentration of FEA over the time for 3 sensors positions. The black lines indicate the time at which the cuts in Figure 5 have been chosen

Figure 5 presents the evolution over the time of the mass concentration of the FEA on the vertical cut.
The propagation of the FEA in the engine bay can be seen on the top pictures, (where the scale is smaller due to the still poor concentration of the FEA). First the area around the FEA outlet will be strongly affected by the presence of FEA. Then the flow disperses more at the front of the engine to finally distribute all over the engine bay. The top area of the engine bay seems always to have a lower concentration than the rest of the compartment. This corresponds to the observations made in Figure 4. However, 1.42s corresponds to the maximum concentration after which the FEA bottle pressure drops together with the velocity at the pipe exit.
After 2s, the FEA is still present everywhere but its concentration is already significantly reduced, finally at time 3.5s, the FEA concentration is again very low (see Figure 5 bottom picture).

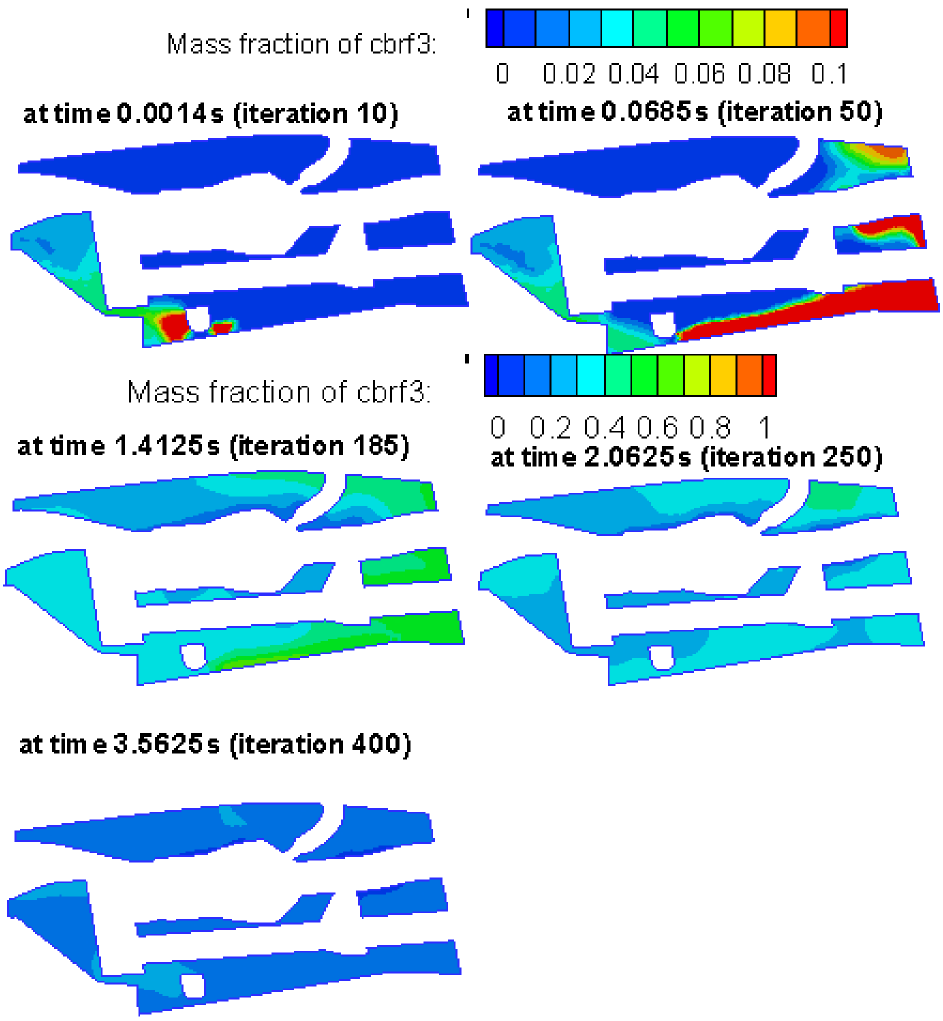



Figure 5: Mass concentration of FEA on a vertical cut through the engine bay and the engine, at 5 different times, the scale is smaller for the two top pictures.

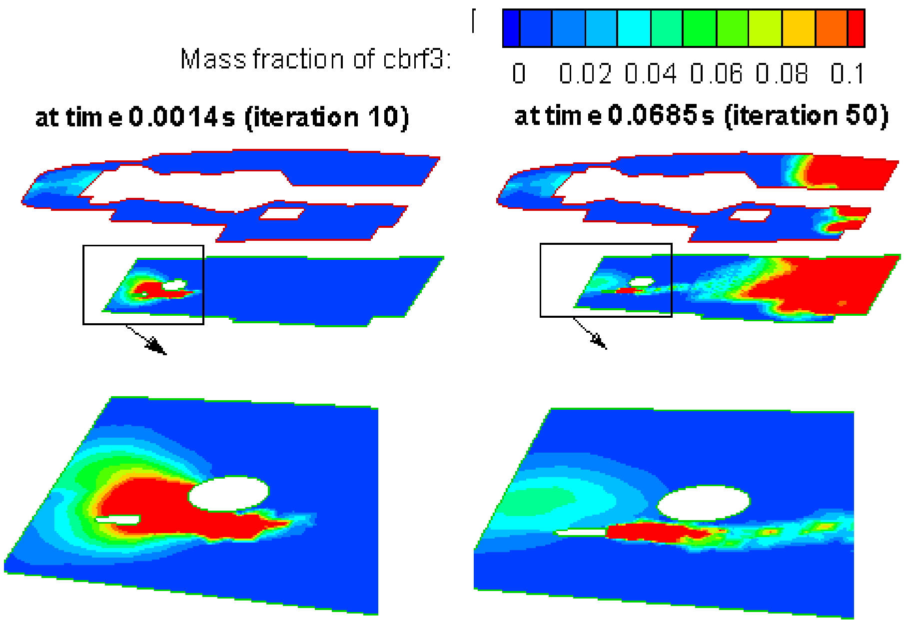



Figure 6: Top picture: mass concentration of FEA on horizontal cuts through the engine bay and the engine, bottom picture: detail of the FEA stream along the fuel filter

In Figure 6, the same process is illustrated as in Figure 5 but at the horizontal cuts. It can be particularly observed (see bottom pictures of Figure 6) that the FEA jet hits the fuel filter and thus the main direction of the FEA stream is slightly deflected. This might be an explanation for a short acceleration of the FEA stream along the fuel filter.

Figure 7 shows the test results of the mass concentration of FEA in percent over the time for 3 sensors: sensor number 1, 3 and 7 compared with the calculation. The plain curves are the results of the CFD calculations, the dashed curves are the results of the tests.

It can be noticed that for the 3 sensors the concentration evolution of FEA over the time is qualitatively similar but the calculated absolute values are too low. Moreover the postponed rise of concentration at sensor 7 is correctly simulated, but the gradient is too small.

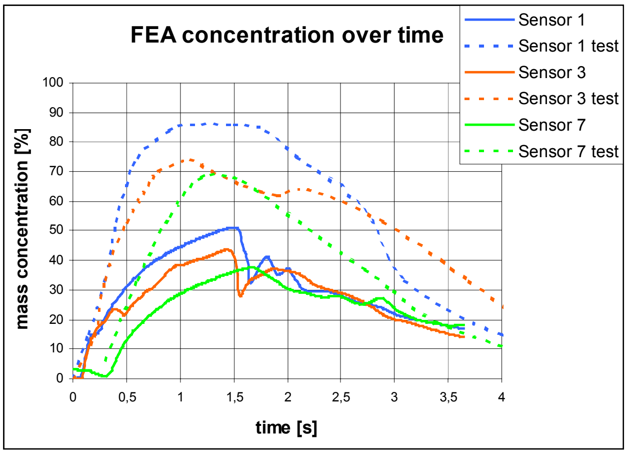



Figure 7: FEA mass concentration in percent at 3 sensor locations (calculation and tests)

The deficit in FEA concentration is caused by the simplifications chosen for this first simulation:

- a relative small mesh, which could now be refined in the critical areas
- constant air velocity assumed on the boundary condition:
  The mass air flow coming in, was taken as constant. In fact, the under pressure created at the outlet through the ejector can be assumed as constant. Although the engine is going down, measurements show that the mass air flow through the engine does not change much in this short period of time. Thus the pressure loss between the air inlet and the outlet at the ejector will stay the same. During the mixing process, pressure will also be lost through the mixing and the diffusing process of the gases (friction) themselves and the pressure difference available for the suction will be reduced. A direct consequence is a smaller mass air flow at the inlet scoops. In order to get the right boundary condition at the air inlet scoops, a profile of velocity should be introduced, based on the reduction of the mass air flow. If the mass air flow is less, automatically the concentration of FEA will increase.
- The calculation was done incompressible, which is not valid for the entire flow, and particularly where the mixing process is important.
- The standard k-ε 2-equation turbulence model shows a weakness in the modelled equation for the dissipation rate. That could be avoided by the utilisation of another turbulence model (realizable k-ε for instance) but on the costs of CPU time.

All these points describe the difficulties encountered and underscore the necessary further efforts for the next steps of the calculation. A better definition of boundary condition is one of the easier parameters to change and will probably have the largest influence on the results.

## Summary and conclusions

This study showed a new and challenging topic, with a complex volume and grid generation, an unsteady process for a certain duration of time (4s), and with partly random boundary conditions, which was CPU time consuming.

However, in the first investigation, it was already a success to simulate the phenomena qualitatively. Of course, a further optimisation and modelisation process is needed in order to show also a validation, which is quantitatively correct. If this can be demonstrated in the future, the CFD tool could be used not only in the design optimisation but also to support the certification process effectively by reducing the number of tests, which are expensive, time consuming, harmful for the environment and give less insight into the 3D flow phenomenon.

## References

[1] "*Meshing Tutorial Manual*", ICEMCFD Engineering, Version 4.0 February 2000

[2] "User's Guide for FLUENT/UNS and RAMPANT", Release 4.0, Vol. 1-3, Fluent Incorporated, April 1986

[3] A. D'Alascio and A. Berthe, "*Industrial use of Computational Fluid Dynamics in the helicopter design process*", Presented at the International 58th Annual Forum of the American Helicopter Society, Montréal, Quebec, Canada, June 11–13, 2002.