30TH EUROPEAN ROTORCRAFT FORUM

Session Flight Mechanics – Aircraft Design

SIMULATION OF FUEL JETTISONING. ANALYSIS OF A COMPLEX FLOW FIELD DURING FLIGHT TEST USING COMPUTATIONAL FLUID DYNAMICS

by

A. D'Alascio EUROCOPTER DEUTSCHLAND GmbH, 81663 München, Germany

> SEPTEMBER 14 - 16, 2004 MARSEILLES FRANCE

SIMULATION OF FUEL JETTISONING. ANALYSIS OF A COMPLEX FLOW FIELD DURING FLIGHT TEST USING COMPUTATIONAL FLUID DYNAMICS

A. D'Alascio

EUROCOPTER DEUTSCHLAND GmbH, 81663 München, Germany

The advances in CFD with respect to physical modelling and numerical accuracy on one side, and the increased computer power on the other, enable the industry to analyse and substantiate the flight test by theoretical tools even for complex flow fields like the fuel dumping problem: a two phases mixture problem in the aft body region of the helicopter.

The fuel jettisoning is an emergency procedure during which the helicopter tank is emptied. This operation must be performed without damaging the helicopter and endangering the crew safety. The main difficulty connected to this type of flight test consists, beside the high costs, in the vague reproducibility, the difficulty to measure the physical parameters and the contamination of the environment, which restricts the allowable amount of tests drastically. The numerical simulation compensates for some of these deficiencies. At relatively low cost most of the local flow parameters of interest can be computed. Furthermore the effect of design modifications and different flight conditions can be analysed and optimised.

The paper gives a general overview about the fuel dumping topic, substantiates the choice of the numerical model used for the CFD simulations and compares the simulation results relative to 4 different flight conditions. The major objective of this analysis has been to support the test engineers in choosing the flight conditions for the fuel jettisoning flight test.

Nomenclature

$\stackrel{\rm g}{L_{ref}}$	Acceleration gravityReference length
m _{Fuel}	- Mass fraction of Fuel = $\rho_{Fuel} v_{Fuel} / (\rho_{Air} v_{Air} + \rho_{Fuel} v_{Fuel})$
• <i>m</i> _{Fuel} Fr	- Fuel mass flow - Froude number = $\rho_{Air}V_{\infty}^2 / \rho_{Fuel}gL_{ref}$
p t	- Static pressure
t t	- Actual time
Δt	- Time step - Volume fraction of air = $V / (V + V)$
, Air V _{Fuel}	- Volume fraction of Fuel = $V_{Fuel}/(V_{Air} + V_{Fuel})$
$V_{Air/Fuel}$	- Volume occupied by the Air/Fuel
$V_{\rm X}$	- Free stream velocity: X-component
V_{Y}	- Free stream velocity: Y-component
VZ	- Free stream velocity: Z-component
V _{AR}	- Descent speed in auto-rotation flight condition

Acronyms

~	
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
EADS-CRC	European Aeronautics Defence and Space – Centre of Research
ECD	EUROCOPTER DEUTSCHLAND
FAR	Flight Aviation Regulation
RANS	Reynolds-Averaged Navier-Stokes
VOF	Volume Of Fluid

30th European Rotorcraft Forum, Marseille, France, September 2004

1 Introduction

The advance in CFD with respect to physical modelling and numerical accuracy on one side, and the increased computer power on the other, is gradually enabling the helicopter industry to simulate more and more complex fluid-dynamic phenomena. As a consequence the spectrum of CFD industrial applications increased during the last few years considerably. EUROCOPTER is already using CFD

- to predict the performance of isolated rotors in hover and forward flight conditions [1], or the drag and lift of isolated fuselages [2];
- to determine aerodynamic loading, acting on fuselage components, which are then applied by the structural department to perform static analysis;
- to support the design department in optimising the shape of helicopter components, *e.g.* engine air intake, rotor head [3];
- to simulate the mixing process of gases, *e.g.* air and an extinguisher gas in the engine bay compartment in case of fire burst out from an engine [4].

EUROCOPTER is newly exploiting the ability of the commercial CFD flow solver FLUENT in simulating multiphase flows, specifically the simulation of a fuel jettisoning process in the aft body region of a large transport helicopter. Here the two phases are air and fuel.

The fuel jettisoning is an emergency procedure during which the helicopter fuel tank is emptied. This operation must be performed without damaging the helicopter and endangering the crew safety.

The main draw back related to this type of flight test consists, beside the high costs, in the difficulty to measure the physical parameters and above all the contamination of the environment, which restricts the allowable amount of tests drastically.

The numerical simulation compensates for some of these deficiencies. At quite low cost most of the local flow parameters of interest can be computed. Furthermore the effect of design modifications and different flight conditions can be analysed and optimised. However, the validation work must be performed accurately in order to "calibrate" the theoretical model. In such a complex physical problem the choice of the numerical model which is better describing the physics is not a trivial task. For this purpose the significant physical quantities need to be identified and the best model available describing the interaction between the two phases must be selected. The selection itself might require a number of numerical additional tests on a simplified geometry. Only then the comparison of the numerical model with the test results can be made. Even when the experiment and the prediction results are not identical after this process, the calculation of "delta effects" is still valuable. It shows in fact the influence of different flight states or design modifications on the fuel dumping process.

This paper gives a general overview about the fuel dumping topic, gives a detailed description of the modelling procedure and shows the simulation results in the flight conditions of interest.

2 The simulation objectives

In compliance with the FAR 29 §1001, if a fuel jettisoning system is installed, it must be safe during all authorised flight regimes and it must be shown that

- the fuel jettisoning system and its operation are free from hazard,
- no hazard results from fuel or fuel vapours impinging on any part of the rotorcraft during fuel jettisoning, and
- the controllability of the helicopter remains satisfactory,

Furthermore the controls of any fuel jettisoning system must be designed to allow flight personnel to safely interrupt fuel jettisoning during any part of the jettisoning operation [5].

The main objective of the CFD simulation was to support the flight department at ECD in choosing the flight conditions to be flown during the fuel jettisoning test campaign. The aerodynamic department had to check if the fuel, during the jettisoning procedure, would interact with warm parts of the helicopter, such as the jet exhausts, or with the tail rotor, thus endangering the safety of the machine and of the crew.

3 The simulation strategy

3.1 Preliminary considerations

The CFD simulation of a single phase external or internal flow is not a trivial task even for an experienced engineer. The crucial aspects which must be considered are various, nevertheless they can be summarised in three basic steps: the meshing strategy of the computational domain, the choice of the numerical scheme and of the turbulence model implemented in the flow solver. Regarding the meshing strategy two basic possibilities are available: structured or unstructured. For highly complex geometry hybrid unstructured meshes are desirable. The choice of the numerical scheme and of the turbulence model is related to several aspects. Of course it depends on the available schemes implemented in the flow solver and on the physical problem itself. The CFD simulation of a 2-phases flow adds to the already mentioned difficulties the additional one of choosing the most appropriate 2^{nd} phase modelling strategy.

For the sake of simplicity all CFD simulations have been run around the isolated helicopter fuselage, *i.e.* neither the main- nor the tail-rotor have been simulated. As a consequence the rotor induced flow on the fuselage has been neglected in all computations. This approximation is acceptable in moderate and high speed forward flight or in descent flight, where the rotor wake is shed away from the fuselage. Nevertheless, considering that the rotor induced flow, added to the main stream, during fuel jettisoning has the effect of pushing the fuel stream away from the helicopter fuselage, the CFD simulation of the isolated fuselage is expected to be conservative with respect to the jettisoning analysis. In fact if the fuel, once released, does not impact the tail boom or the tail rotor, the addition of the main rotor induced flow would increase the distance between the fluid and the fuselage components.

3.2 The flight conditions

The three flight conditions of Table 1, specified by the flight test department at ECD, differing only from the descent velocity V_Z , have been simulated. The horizontal velocity is V_X=60 Knots in all selected test cases. All flight conditions are symmetrical, thus $V_{Y}=0$. As far as the descent velocity is concerned, the range of interest was between 0 and 75% of the auto-rotation descent speed VAR at dive speed. The fuel mass flow \dot{m}_{Fuel} dumped from the helicopter tank through the exhaust section has been considered constant, although its value decreases, while emptying the tank, according to the Torricelli law. The \dot{m}_{Fuel} value chosen for the CFD computations is the nominal value fixed by a valve placed at the two pipe-line entrances (see also Figure 2). It represents therefore the maximum reachable mass flow obtained when the fuel tank is full.

Case	$\mathbf{V}_{\mathbf{Z}}$
TC1	$V_{Z1} = 0.75 \cdot V_{AR}$
TC2	V _{Z1} /2
TC3	V _{Z1} /4

 Table 1: Free stream conditions for the three selected test cases.

The fuel jettisoning starts when the helicopter flies steadily in the specified flight conditions.

3.3 The mesh generation

Starting from an already existing surface model of the heavy transport helicopter fuselage, the complete fuel dump pipeline, comprising of the two dump outlets, the pipe lines and the fuel dump exhaust section, has been integrated in the existing fuselage model, thus obtaining the surface geometry of Figure 1 and Figure 2. The first figure shows a back view of the complete surface model, whereas the second a zoom on the fuel dump pipeline.



Figure 1: Surface model of the heavy transport helicopter isolated fuselage.



Figure 2: Particular of the fuel dump pipeline.

In order to correctly define the boundary condition to be applied at the dump exhaust section (see Figure 2), *-fuel inlet section* for the CFD computational domain–, a short region inside the fuel dump pipeline has been meshed. As a consequence the fuel dump exhaust section has been moved inside the pipe. In this way the direction of the fuel flow entering the computational domain through the dump exhaust section is univocally defined by the pipe axis.

For the generation of the volume grid around the helicopter fuselage the unstructured methodology was preferred to the structured one, because it allows to generate a volume grid around highly complex surfaces in shorter time. A number of meshes have been generated by EADS-CRC in collaboration with ECD by using the unstructured grid generator CENTAUR. The dimensions of the first and the last mesh are listed in Table 2.

	Nr. of cells	Nr. of nodes
First mesh	1682753	325617
Final mesh	525694	136272

Figure 3 shows the middle plane of the first unstructured mesh, whereas Figure 4 shows the same plane in the final mesh. It can be noticed that, although the number of cells in the final mesh is definitely smaller than the one in the first mesh, the region behind the helicopter fuselage, where the fuel is jettisoned, is definitely finer discretised in the final mesh.



Figure 3: Middle plane of the first unstructured mesh.



Figure 4: Middle plane of the final unstructured mesh.

Figure 5 shows a plane parallel to the middle plane of Figure 4 through the fuel dump exhaust section axis. The small quadrant on the left shows a zoom in the exhaust pipe region and the small quadrant on the right shows the particular of the mesh inside the exhaust pipe. The mesh is a pure unstructured grid composed by tetrahedral elements. No additional effort has been made to generate a prism layer around the fuselage, necessary to model the boundary layer correctly. The CFD simulation must be accurate enough to predict the velocity field induced by the fuselage aft-body region. Therefore the final grid has been drastically coarsened in the fuselage front region and around the engine cowling while increasing the grid resolution in the aft-body region.



Figure 5: Final unstructured mesh: plane through the fuel dump exhaust section.

3.4 The flow solver

The incompressible version of the Navier-Stokes commercial flow solver FLUENT ver. 6.1 has been run in steady and unsteady modes. For each flight condition of Table 1 a first steady computation has been made in order to reach the steady state at which the fuel dump was started. The steady computation has been considered converged when the continuity equation residual decreased of 4-5 orders of magnitude and the global loads reached a constant value. The steady solution has been applied as the initial condition to the following unsteady computation. In both calculations, steady and unsteady, the multiphase option of FLUENT had been activated in order to model the two phases, air and fuel. In fact, even though the steady computation solves a single phase problem, it was decided to activate also the set of equations modelling the second phase, in order to have all primitive variables correctly defined in the initial condition of the subsequent unsteady multiphase computation. All computations are fully turbulent: no laminar region has been accounted for. The standard k-E turbulence model with the standard wall functions for the near wall treatment has been applied. The gravity force term of the momentum equations has been activated, being the Froude characteristic number of this problem of the order of $Fr \cdot L_{ref} \approx 0.1$

3.4.1 The choice of the 2-phase model [6]

FLUENT implements two approaches for the numerical calculation of multiphase flows: the Euler-Lagrange approach and the Euler-Euler approach. The Lagrangian discrete phase model in FLUENT follows the **Euler-Lagrange approach**. The fluid phase is treated as a continuum by solving

the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles, bubbles, or droplets through the calculated flow field. The dispersed phase can exchange momentum, mass, and energy with the fluid phase. A fundamental assumption made in this model is that the dispersed second phase occupies a low volume fraction, even though high mass loading $\dot{m}_{Particles} \ge \dot{m}_{Fluid}$ is acceptable. The particle or droplet trajectories are computed individually at specified intervals during the fluid phase calculation.

In the Euler-Euler approach, the different mathematically phases treated are as interpenetrating continua. Since the volume of a phase cannot be occupied by the other phases, the concept of phasic volume fraction is introduced. These volume fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of equations, which have similar structure for all phases. These equations are closed by providing constitutive relations that are obtained from empirical information, or, in the case of granular flows, by application of kinetic theory.

In FLUENT, three different Euler-Euler multiphase models are available: the volume of fluid (VOF) model, the mixture model, and the Eulerian model.

The **VOF model** is a surface-tracking technique applied to a fixed Eulerian mesh. It is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest. In the VOF model, a single set of momentum equations is shared by the fluids, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain.

The **mixture model** is designed for two or more phases (fluid or particulate). As in the Eulerian model, the phases are treated as interpenetrating continua. The mixture model solves for the mixture momentum equation and prescribes relative velocities to describe the dispersed phases.

The **Eulerian model** is the most complex of the multiphase models in FLUENT. It solves a set of n momentum and continuity equations for each phase. The coupling is achieved through the pressure and the inter-phase exchange coefficients. The manner in which this coupling is handled depends upon the type of phases involved; granular (fluid-solid) flows are handled differently than non-granular (fluid-fluid) flows.

The fuel, jettisoned approximately at a velocity of 1m/s, is convected downstream by the air flow at a speed of about 30m/s. The difference of velocity between air and fuel generates high friction between

the two media, which breaks immediately the gasliquid interface, thus dispersing the liquid in the gas medium. Nevertheless, as the liquid phase is injected in the computational domain as a continuum, the Euler-Euler approach had to be selected. The simplest VOF and the most accurate Euler model were tested on a simpler geometry. The use of the VOF model, most suited for low velocity flows, in this flight condition requires a very fine discretisation of the computational domain in the region where the fuel is dumped and a very small time step for the unsteady run. Therefore it was preferred to use the more sophisticated Euler model which allows for a coarser grid and a higher time step. With the above mentioned setting FLUENT solves for each time step a set of 12 differential equations: the continuity equation for the mixture, 3-momentum and 2-turbulence equations for both phases and the volume equation for the second phase.

Model	Solver:	3D, Steady/Unsteady, Implicit, Cell-
		based
	Multiphase:	Eulerian
	Viscous:	Standard k-E with wall function
Phases	Phase 1:	Air
	Phase 2:	Kerosen
Operating		Gravity Force activated
Condition		

Table 3: FLUENT parameter setting.

3.4.2 The boundary and initial conditions

The boundary conditions for the steady and the unsteady simulation are reported in Table 4.

Boundary section	Steady run	Unsteady run	
Fuel dump exhaust section	$\dot{m}_{Fuel} = const \ [kg / s]$	$\dot{m}_{Fuel} = 0 [kg / s]$	
Inflow section at far field	V∞ [m/s] See Table 1		
Outflow section at far field	P _{out} =0 [Hpa]		
Wall	V _{wall} =0	[m/s]	

Table 4: Definition of the boundary conditions for the unsteady and the steady initialising runs.

Once computed the steady solution around the helicopter fuselage, the boundary condition at the fuel dump exhaust section has been modified according to the description of Table 4, the unsteady terms of the N.-S. equations have been activated and the unsteady run has been started by applying the steady solution as initial condition. It must be mentioned that an initial condition of a multiphase unsteady problem in FLUENT must be so defined that both phases exist in the computational domain. It is not sufficient that the second phase be injected through a boundary surface in the control volume, it must be present in the volume already. For this reason at the initial time of the unsteady computation a *domain patch* was defined, by filling up with the liquid phase a small portion of the pipeline adjacent to the fuel exhaust section.

The physical time step of the unsteady computation was set to $\Delta t=0.0025[s]$.

4 Flow analysis

The chapter presents the CFD prediction results around the large transport helicopter. In the first section the steady and unsteady results of the test case TC1, *i.e.* isolated fuselage in level flight conditions, are exhaustively presented. The second section is dedicated to the comparison of the fuel dump unsteady results in the four flight conditions considered. A short paragraph showing a qualitative comparison between computation and flight tests closes the chapter.

4.1 The isolated fuselage in medium descent flight (TC2).

Figure 6 shows the pressure coefficient distribution on the helicopter fuselage. The typical behaviour characterised by a stagnation region on the fuselage nose, followed by a flow acceleration on the fuselage central part and a subsequent deceleration on the rear part, can be observed.



Figure 6: Pressure coefficient distribution on the helicopter fuselage (TC2 – initial condition).

Figure 7 shows the fuel mass fraction on a vertical plane through the fuel exhaust pipe and on the fuselage surface. Four time shots have been selected: t=0.75, 1.0, 1.2 and 1.35. It can be seen that the solutions, in terms of fuel mass fraction, at t=1.2[s] and at t=1.35[s] are practically identical. In fact in all test cases run it has been observed, that the field solution reaches a "steady state" after one second of fuel jettisoning. The same occurrence has been observed during flight tests. This can be justified by the fact, that in one second, due to the flight velocity of about 30[m/s], the ejected fuel is convected approximately 30 meters downstream of the helicopter back door. Considering that the helicopter tail is about 8 meters long, each fuel particle ejected at a time t_0 might impact the helicopter tail, empennage or tail rotor during the first 0.3[s] from the release instant t_0 . In one second therefore the position of the fuel cloud behind the helicopter fuselage seems to have reached a constant state in terms of fuel mass distribution and spatial position.



Figure 7: Fuel mass fraction on a vertical plane through the fuel exhaust pipe (TC1 – unsteady run).

It must be highlighted that scale of the legend relative to the fuel mass fraction distribution on the fuselage surface (Figure 7-right column) has been restricted to the range [0: 0.1] in order to show the fuselage wetted surface even for fuel mass fraction smaller that 10%.

4.2 Comparison of the fuel jettisoning results

Figure 8 shows a comparison between the CFD results obtained in the three flight conditions of Table 1 in terms of fuel mass fraction on a vertical plane through the exhaust pipe. The "steady solution" reached during each unsteady computation after one and a half seconds is here depicted for each test case. Figure 9 shows a snapshot of the fuel jettisoning flight test in the flight conditions of TC3: light descent. The CFD solution is qualitatively in good agreement with the flight tests. As expected the numerical predictions appears to be conservative with respect to the flight test results. In fact the mean fluid flow depicted in the flight test snapshot of Figure 9 is convected downstream almost parallel to the tail boom axis, whereas the CFD prediction of Figure 8-bottom shows a small inclination towards the tail boom. This small difference is most probably due to the lack of the main rotor induced flow in the CFD computation, but also to an incorrect prediction of the separation region, which is known to take place at the back door of the helicopter fuselage.



Figure 8: Comparison between the CFD results relative to the 3 test cases. Fuel mass fraction on a vertical plane through the fuel exhaust pipe.



Figure 9: Snapshot of the fuel jettisoning flight test in the flight conditions of TC2.

Figure 10 shows again the comparison between the three CFD results in terms of fuel mass fraction. The pictures on the left depict the patterns on transversal planes behind the fuselage backdoor, whereas the pictures on the right show the distribution on the fuselage surface. Also here the legend scale has been adjusted to show the wetted surface even for fuel mass fraction smaller that 10%.



Figure 10: Comparison between the CFD results relative to the 3 test cases. Patterns of the fuel mass fraction behind the fuselage (left) and on the fuselage (right).

From a more detailed analysis of Figure 9 and Figure 10 it can be seen that:

- in none of the selected flight conditions the fuel vapour meets hot surfaces such as the engine ejectors;
- in the test case TC1 the fuel jettisoned flies through the tail-rotor. This might be an hazardous manoeuvre, therefore the flight condition TC1 was not selected for the fuel dump flight test campaign.
- No considerable difference has been observed between the results relative to the flight condition TC2 and TC3;
- the comparison between flight test and computation in the flight condition of TC2 is qualitatively fairly good.

5 Conclusions and outlook

The paper has presented the CFD analysis of the fuel jettisoning problem around the isolated fuselage of a large transport helicopter. The choice of the most suited multiphase model, among the available ones implemented in the commercial flow solver FLUENT, has been explained. The comparison between the CFD results obtained for the selected flight conditions has been shown, as well as a qualitative comparison between the CFD computation and the flight test results. The prediction results show qualitatively good agreement with the flight test measurements in the selected flight condition TC3.

In conclusion, the application of CFD analysis has allowed the flight test department at ECD to select the flight conditions to be flown during the fuel jettisoning test campaign. The aerodynamic simulation has allowed the flight test department to reduce the risks during flight.

The same methodology can be applied to optimise the fuel dump pipeline (particularly its last segment) with the objective of increasing the distance between the fuel free jet and the helicopter tail-boom. In fact a CFD analysis around different geometrical configurations carried out, by keeping both flight condition and flow solver numerical parameters unchanged, would highlight only the effect of the geometry modification on the flow field thus showing the best configuration.

References

- A. D'Alascio, C. Castellin, M. Costes and K. Pahlke, "Aerodynamics of Helicopter. Application of the Navier-Stokes Codes developed in the Framework of the joined German/French CFD Research Program CHANCE", Presented at the CEAS Aerospace Aerodynamics Research Conference, Cambridge, UK, June 2002.
- [2] A. D'Alascio, A. Berthe and F. Le Chuiton, "Application of CFD to the fuselage aerodynamics of the EC145 helicopter. Prediction of unsteady phenomena and of the time averaged influence of the main rotor", Proceedings of the 29th European Rotorcraft Forum, Friedrichshafen, Germany, September 2003.
- [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, 2002.
- [4] A. Berthe and A. D'Alascio, "CFD simulation of fire extinguishing system: two gases mixture inside the engine bay", Proceedings of the 29th European Rotorcraft Forum, Friedrichshafen, Germany, September 2003.
- [5] Amdt. 29-26, 53 FR34218, Sept. 2,1988
- [6] *"FLUENT 6.1 User's Guide"*, Fluent Incorporated, January 2003