31st EUROPEAN ROTORCRAFT FORUM

Session CFD Methods for Complete Helicopter
Paper #69

OVERVIEW OF RESULTS OBTAINED DURING THE 6-YEAR FRENCH-GERMAN CHANCE PROJECT

K. Pahlke*, M. Costes, A. D'Alascio***, C. Castellin****, A. Altmikus*****

*DLR, Braunschweig, Germany
**ONERA, Châtillon, France
***ECD, Ottobrunn, Germany
****EC, Marignane, France
*****University of Stuttgart, currently at ECD, Ottobrunn, Germany

Presented at the European Rotorcraft Forum, 13-16 Sept. 2005, Florence, Italy.

OVERVIEW OF RESULTS OBTAINED DURING THE 6-YEAR FRENCH-GERMAN CHANCE PROJECT

K. Pahlke*, M. Costes, A. D'Alascio***, C. Castellin****, A. Altmikus*****

*DLR, Braunschweig, Germany
**ONERA, Châtillon, France
***ECD, Ottobrunn, Germany
****EC, Marignane, France
*****University of Stuttgart, currently at ECD, Ottobrunn, Germany

Abstract

The present paper gives an overview of the CHANCE research project (partly supported by the French DPAC and DGA and the German BMWA) which was started more than 6 years ago between the German and French Aerospace Research Centres DLR and ONERA, the University of Stuttgart and the two National Helicopter Manufacturers, Eurocopter and Eurocopter Deutschland. The objective of the project was to develop and validate CFD tools for computing the aerodynamics of the complete helicopter, accounting for the blade elasticity by coupling with blade dynamics. The validation activity of the flow solvers was achieved through intermediate stages of increasing geometry and flow modelling complexity, starting from an isolated rotor in hover, and concluding with the time-accurate simulation of a complete helicopter configuration in forward-flight. All along the research program the updated versions of the CFD codes were systematically delivered to Industry. This approach was chosen to speed up the transfer of capabilities to industry and check early enough that the products meet the expectations for applicability in the industrial environment of Eurocopter.

Introduction

Among the various configurations encountered in aerospace applications, rotorcrafts are certainly among the most complex to handle from the numerical simulation point of view. Indeed, as far as aerodynamics is concerned, helicopters combine transonic and low-speed unsteady flows, high angles of attack, bodies in relative motion, strong wake interactions, unsteady transonic regions at the tip of the advancing blade and separated areas on the retreating one. Furthermore, contrary to fixed-wing aircraft, the rotor motion is the result of a coupling between aerodynamic and inertial forces and moments, because of the articulated hub or concentrated softness, so that the blade motion is part of the solution for each flight configuration. Finally, since blades are lifting surfaces of high aspect ratio and that most of the important aerodynamic phenomena occur at the tip where the dynamic pressure is the highest, blade deformations are significant and must be correctly simulated. As a result, the application of Computational Fluid Dynamics in the rotorcraft industry has some delay if compared to aircraft and propeller manufacturers.

On the other hand, because of their inherent complexity, rotorcraft applications are good candidates for the development and application of new CFD techniques not necessarily needed for simpler configurations. This is indeed the case for the Chimera technique. The overlapping grid strategy is almost mandatory to represent the blades motion around the fuselage [1]. Moreover accurate wake resolution and wake conservation within Euler/RANS solutions has been a great concern since the early developments of rotorcraft CFD, much in advance with respect to fixed wing studies on this item [2], [3], [4], [5], [6], [7]. Consequently, this shows that the CFD community can draw significant benefits from the research experience existing in rotorcraft applications.

This status led ONERA and DLR, together with University of Stuttgart, Eurocopter and Eurocopter Deutschland to define a common 6-years research program called CHANCE (Complete Helicopter AdvaNced Computational Environment) in order to intensify the development of CFD capabilities for rotorcraft applications in both countries. The present paper aims at giving an overview of this program and showing some significant results which were obtained by the various partners in the course of the project.

Description of CHANCE

Status of CFD before CHANCE

Prior to CHANCE, the development and application of CFD for rotorcraft applications had been an ongoing effort both in Germany and France for more than two decades. Although a large part of work had been conducted in the frame of national research programs, common activities started at the end of the eighties through EU-funded Brite-Euram projects. Finally the bi-lateral cooperation in the field of CFD for rotorcraft between DLR and ONERA was strongly encouraged by the creation of the single French-German helicopter manufacturer Eurocopter. Such a bi-lateral cooperation allowed getting a more detailed knowledge of the methodologies used in both countries and to have more detailed comparisons of numerical techniques as well as computed results. This was a very helpful preliminary step in the preparation of CHANCE.

In terms of know-how, the CFD capabilities at the beginning of CHANCE for helicopter applications were the following [8]:

- RANS computation of isolated rotor in hover, using algebraic turbulence models,
- RANS computation of isolated fuselage, using algebraic or 1- or 2-transport equations turbulence models,
- Euler computation for isolated rotor in forward flight for rigid or flexible blades,
- Euler computation of unsteady rotor + fuselage interaction for very simplified geometries using Chimera,
- Euler computation of quasi-steady rotor + fuselage interaction using an actuator disk approach,
- Weak coupling with blade dynamics for an isolated rotor using a full-potential model for the blade aerodynamics.

Nevertheless, such an experience does not imply that all these capabilities were either available or fully operational in both countries and that they could be combined for use in an industrial project. Indeed, the various techniques were generally implemented in different CFD codes or versions of software and therefore combining these functionalities for dealing with a complex case was generally not possible. Furthermore, these methods were essentially used by the research centres or universities, while the use of CFD by industry was less advanced and generally limited to the application of commercial software for analysing specific helicopter components [9].

Objectives of CHANCE

The CHANCE program aimed at the development and validation of CFD tools for simulating the flow field around the complete helicopter, including the coupling with rotor trim and blade dynamics. To reach such an ambitious objective all aspects of the lifetime of numerical methods in CFD need to be covered: algorithmic development, software coding and verification for simplified test cases, validation for actual helicopter geometries, including the generation of grids adapted to the phenomena which are to be simulated, the transfer, installation, checking and application of the method in the industrial environment and the software versions and quality management.

In order to keep the project objectives realistic, the most challenging helicopter flight conditions were excluded from the test case validation matrix, *i.e.* the ones causing BVI, tail shake and dynamic stall. Indeed, both the limitations of numerical techniques as well as of physical modelling could not let us foresee the possibility of tackling these phenomena during the course of the project. It was rather decided to leave the analysis of these difficult points to subsequent and more specific studies. Nevertheless, it was also agreed that the CFD methods should be capable of computing (in terms of stability and robustness) these configurations, whatever the accuracy of the simulation might be.

The duration of the CHANCE project, initially planned for 6 years, finally extended up to 6.5 years. This can be explained by the difficulty of accurately scheduling this research activity for such a long period of time. The work program was carefully built step by step, in common between the partners. Schematically, the research centres and universities were responsible for the development and validation of the software, whereas industry was in charge of integrating the methodologies in its own environment and testing the software on industrial problems. All along the project, intermediate versions of software were transferred to industry, together with portability test cases in order to make sure that the installation worked correctly. This was felt necessary because research centres and universities are generally using supercomputers with vector architecture, while the industrial computations are run on parallel scalar workstations.

Numerical methods

During the CHANCE project, different CFD methods were used in France and Germany. Although this was felt as a weakness, the constraint could not be avoided. There were indeed very good reasons for that. First, the CFD methods used in both countries are multi-applications not specific to helicopters, for obvious efficiency reasons; second, most of the cost to develop these computer codes was covered by national funding, both in France and Germany. Therefore, in spite of the unique position of the harmonised helicopter research groups between France and Germany at the beginning of the project driven by the single French-German helicopter manufacturer Eurocopter, all the background knowledge and software development could not be put in common. Nevertheless, this restriction did not have only drawbacks; in fact the comparison of different implementations of numerical techniques as well as the comparison of flow solutions highlighted errors and were helpful in the debugging process and helped to increase confidence in CFD at industry.

In France, the CFD software used for the CHANCE project is *elsA* [10]. The *elsA* solver is the multi-application object oriented aerodynamic code of which the development started at ONERA in 1997. It is based on a cell-centred finite volume technique for structured multi-blocks meshes and includes a wide range of numerical techniques as well as physical models in order to simulate the flow-field around realistic aerospace configurations from the low subsonic to the hypersonic regime. The domain of application includes fixed wing, rotary wing, turbo machinery, space launcher and missile configurations. In the present activity, the spatial discretisation used is the standard Jameson's second-order centred scheme with explicit artificial viscosity terms using second and fourth differences. A backward Euler explicit time integration technique is applied with a 4-stage Runge-Kutta algorithm, together with implicit residual smoothing. For steady-state problems, local time-stepping and multigrid acceleration techniques are applied to speed-up convergence. Low-Mach number preconditioning techniques are also available for very low speed flows. For unsteady applications, either the dual-time stepping technique or the Gear time-integration scheme is generally used for an implicit formulation of the problem. The former allows using all the techniques developed for steady-state solutions during the internal sub-iterations, while the latter uses Newton sub-iterations at each time step to reach convergence. The implicit system is solved using LU decomposition.

In Germany, the CFD method used in CHANCE is FLOWer [11], [12]. FLOWer is a portable software system and can be run on a large variety of computers with high efficiency. The domain of application is similar to that of *elsA* although FLOWer has not been adapted for hypersonic flow. FLOWer was developed by DLR, and solves the compressible Euler/Navier-Stokes equations for structured multi-block meshes, using a finite-volume scheme with 2nd order centred space discretisation and explicit artificial viscosity, and a 5-stage 2nd-order Runge-Kutta explicit method for advancing in time. During the run of CHANCE the cell-centred space discretisation was introduced in FLOWer for all functionalities in addition to the already existing cell-vertex discretisation. Several upwind schemes are available and a specific implicit treatment for the robust time integration of multi-equation turbulence models was developed. The development of a fully general chimera technique was achieved during the run of the project. All the convergence acceleration techniques such as local time-stepping, implicit residual smoothing and multigrid can be used for steady flows. For unsteady applications, the dual time-stepping technique is used so that, for each physical time step, internal iterations are performed using the same techniques as for steady flows in order to converge towards the time-accurate solution.

The rotor dynamics code used in the frame of CHANCE is the HOST code from Eurocopter [13]. This helicopter comprehensive analysis is the main simulation tool used by Eurocopter for aeromechanics and flight dynamics simulation. It includes the modelling of the various components of the helicopter and can be applied for trim, simulation in the time domain and stability analysis. Either the full helicopter or isolated components such as the main rotor can be computed. The rotor aerodynamics is basically given by blade element theory, using the 2D airfoil tables for providing the sectional forces and moments which are applied at the quarter-chord line of the blade. For computing the induced velocities, several inflow models are available such as the Meijer-Drees analytic model or the METAR vortex-lattice method. The blade structure uses the 3-degrees of freedom beam theory with modal decomposition in order to reduce the number of unknowns when solving the Lagrange's equations. Since the introduction of

HOST started lately at the DLR Institute of Aerodynamics and Flow Technology most of the fluid-structure coupling activities were carried out at DLR using the comprehensive rotor simulation code S4 [14] of DLR FT as rotor dynamics code.

Steps of the research

The various steps of the CHANCE project were carefully defined in order to progressively consider more and more complex configurations and functionalities towards the final application.

To begin with, isolated components of the helicopter were considered, such as the main rotor (in hover and forward flight) and the fuselage. First, this allowed to integrate all the existing know-how into a general multi-purpose CFD software, and then to extend it and develop new functionalities necessary for dealing with helicopter applications. This was more particularly the case for performing viscous computations of the rotor in forward flight, for which the various numerical techniques related to ALE, for describing the blade motion and deformation, had to be adapted in order to efficiently and accurately compute the deforming blade with azimuth.

As far as coupling with blade dynamics is concerned, two approaches were developed, called weak and strong coupling. The weak coupling technique, valid for periodic flight conditions, takes advantage of the flow periodicity to exchange the blade loads and dynamics between HOST and CFD at each rotor revolution. The rotor is trimmed using 2D airfoil tables corrected by CFD loads and moments computed during the previous coupling iteration, so that, at convergence the trim corresponds to the CFD distribution of loads and moments all over the rotor disk. Contrary to that, the strong coupling technique is time accurate from the very beginning of the coupling process and exchanges information between HOST and CFD at each step of a time marching process until a periodic state is obtained. The drawback of this approach is that the rotor trim cannot be reached directly. Indirect ways of reaching the rotor trim were thus developed.

For computing the complete helicopter, two approaches were adopted. First, a time-averaged simulation of the helicopter in steady flight conditions was developed using an actuator disk approach. In this case, the rotor blades are replaced by a distribution of time-averaged forces on the rotor disk. These forces may be either uniform or varying with azimuth and radial position to better represent the non-uniform downwash which appears below the rotor disk. In that case, the prescribed force distribution is obtained from blade element theory, for example by using HOST. This first approach could be made operational early enough in the project in order to allow industry to apply it to realistic configurations and investigate specific helicopter problems. The second and last approach developed for computing the complete helicopter is the full unsteady simulation taking into account the blades motion relative to the fuselage. Such an application required to combine several advanced numerical techniques such as the chimera overset grid approach and the ALE formulation for deforming bodies. Because of the complexity of such kind of approach, great care was put in the efficiency of the numerical techniques applied in order to make the computation feasible.

Validation and delivery to industry

All along the project, code validation was completed for the various functionalities mentioned above, in order to make sure that the developed numerical techniques are appropriate and work correctly. A list of test cases was selected under the responsibility of the industrial partners for all these configurations considered. Among these, the research centres and universities mainly concentrated on simpler but fairly well-documented test cases acquired in wind-tunnels, whereas industry applied the CFD codes to their in-house products, thus verifying the advantages which CFD can bring in a product development process.

Regular delivery of the CFD methods to industry was made all along the project. Each time a new functionality was available, a portability test case was attached to the code delivery for checking that similar results are obtained. This allowed industry to apply the CFD methods to their in-house activities early enough in the project.

Examples of Results

Introduction

Typical examples of the results obtained within CHANCE are presented in this part. They aim at illustrating the main activities completed during CHANCE. Only results that were published in a journal, on a national or an international conference until Sept. 05 are shown in this chapter. The test cases will be presented in an order with increasing modelling complexity, i.e. starting with isolated fuselages, going to the quasi-steady simulation of rotor-fuselage configurations in which the rotors are modelled as actuator disks and finally to the time-accurate simulation of a main rotor – fuselage – tail rotor – model support configuration.

<u>DGV fuselage [15]</u>

The reduction of fuselage drag is an important aspect of helicopter aerodynamics since this part of the helicopter represents a significant part of the drag of the complete rotorcraft, especially at high-speed. CFD can thus be of great help in order to design optimised fuselage shapes, but prior to this the computations must be deeply validated by comparison with experimental data. An example is given here for the DGV fuselage which was tested in the ONERA F1 wind-tunnel.

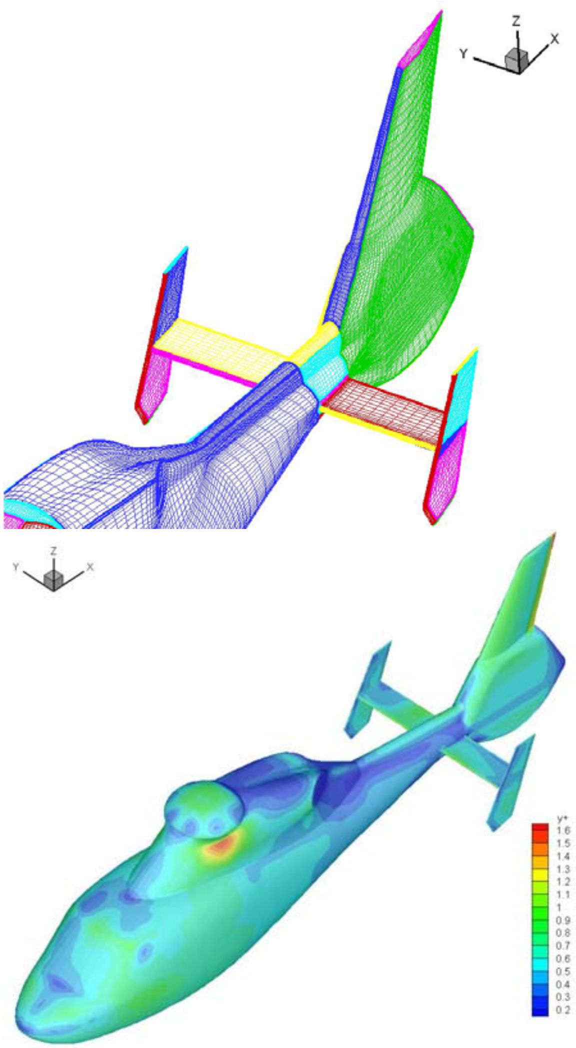

Figure 1 View of the surface grid near the tail and of the y+ distribution

The computation [15] around this kind of complex geometry requires a significant effort for generating grids of sufficient quality. The present grid includes about 5 million points, distributed over more than 70 blocks (Figure 1). The y^+ could be maintained below 1 almost everywhere except just below the rotor hub where large surface curvature renders the meshing more complex. The computation presented here was performed assuming fully turbulent flow, using Smith's two transport equations k-l turbulence model. The pressure distributions along the top and bottom centrelines of the fuselage are presented in Figure 2 for a free-stream Mach number of 0.235 and a free-stream Reynolds number of 15 million. They correlate well with experiment, the main difference coming from the pressure variations in the vicinity of the strut which was not accounted for in the computations. Nevertheless, an overestimation of the experimental drag of about 15% is obtained, showing that much finer grids are required to accurately simulate the important flow features around such kind of complex geometries.

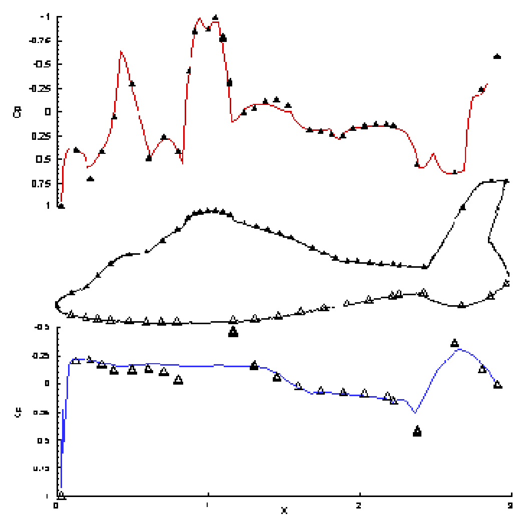

Figure 2 Pressure distribution along top and bottom centrelines

<u>Isolated BO105 Wind tunnel model [16]</u>

Generation of high quality structured grids around complex geometries is the most time and effort consuming step in the numerical simulation process. Overlapping grid techniques offer an attractive alternative to the classical multi block approach by breaking down complex configurations into a number of simple components, generating structured grids around each component individually, and interpolating the solution between the component grids during the solution process using a background grid which partially overlaps them.

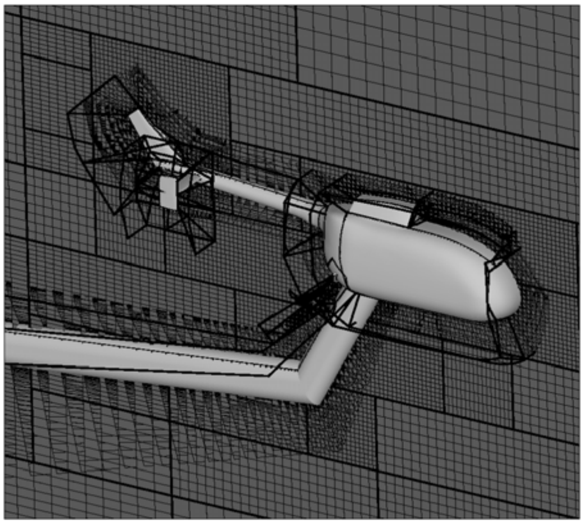

Figure 3 Chimera grid system with Cartesian background grid around BO105 wind tunnel model

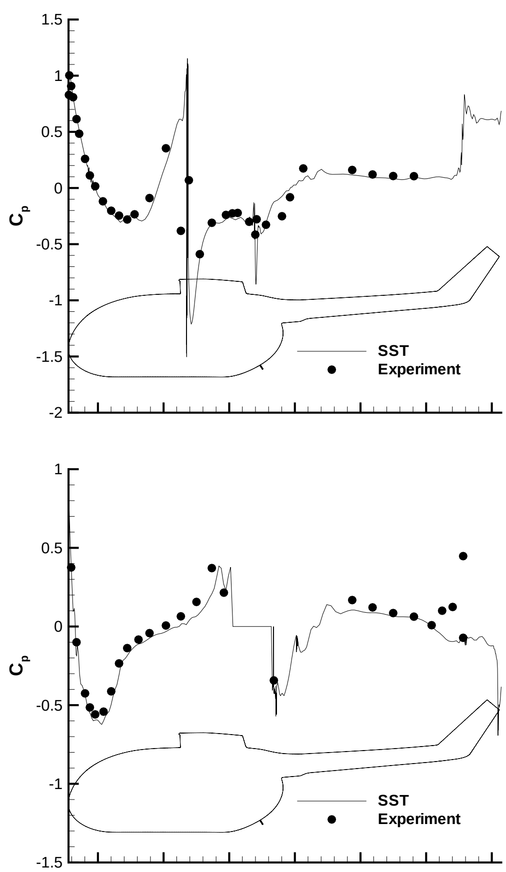

Figure 4 Pressure coefficient at symmetry plane for α=-1.87°(top: upper side, bottom: lower side)

However, proper overlap between the grids is essential for the stability and accuracy of the method. The interpolation of the data should take place far from high gradients (boundary layers, shocks, ...) otherwise the solution is contaminated by large interpolation errors, which may prevent the convergence of the solution. To guarantee accurate interpolation, the overlapping grids must have comparable resolutions within the overlap regions. Satisfaction of the above conditions using one of the component grids (usually of the largest dimensions) as a background grid places additional constraints on grid generation, and may result in poor quality grids in many cases. In addition, considerable amount of time and effort is usually required before a moderate quality functioning Chimera grid can be obtained. Thus, Chimera application becomes less attractive in practice when complex three dimensional configurations are involved, for which the technique is supposed to be most beneficial.

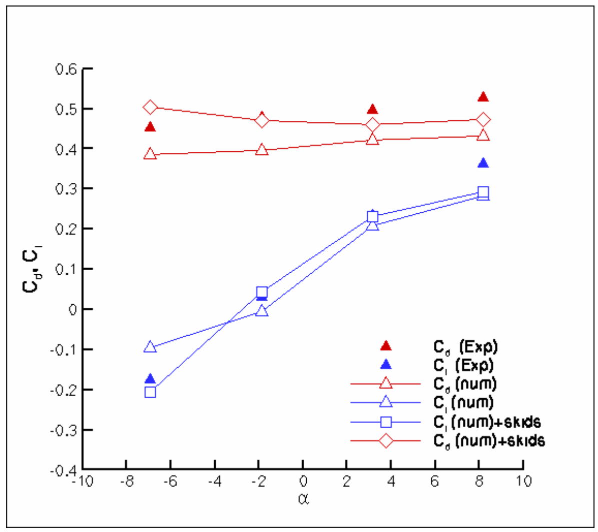

Figure 5 Drag and lift coefficient as a function of pitch angle (BO105)

In this paragraph an alternative approach to transfer the data between the child grids is applied. The approach replaces conventional boundary fitted background grids by a Cartesian grid. This approach has the following advantages. Firstly, a Cartesian grid requires negligible human and computational effort. Secondly, identical grid resolutions can be achieved in the overlap regions. Thirdly, Cartesian grids offer ideal numerical characteristics (no cross diffusion). Block local refinement is employed to ensure matching grid resolutions in the overlap zones only, thus keeping the number of grid points within acceptable limits (see Figure 3). A comparison of the predicted [16] and measured [17] pressure distribution in the centreline for M=0.136, α=-1.87o and Re=8.51 Mill. is presented in Figure 4 showing a good agreement between prediction and experiment. The computation was carried out with the k-ω SST turbulence model. Figure 5 presents the polar of the global forces. In addition to the global forces of the bare fuselage, the values for the fuselage with skids are shown which compare considerably better with the experimental data which were also obtained with mounted skids.

The EC145 isolated fuselage [18]

The EC145 twin engine helicopter, depicted in Figure 6, is equipped with a 4-bladed main rotor, a conventional 2-bladed tail rotor and two landing skids. The shape of the rear side of the fuselage is formed blunt by a backdoor allowing the loading from the back. Such configuration makes the EC145 particularly suited to security and rescue missions, but it definitely complicates the flow field structure, and in consequence the aerodynamic predictions and analysis. In fact the steep junction between the fuselage body and the tail boom is responsible for a separated flow region which might give birth to unsteady vortex structures which are shed downstream (von Kármán vortices). In this case the use of the most sophisticated turbulence models is mandatory to accurately predict the fuselage drag. In addition, when unsteady phenomena occur, the solution of the URANS equations is necessary.

Figure 6 EC145 helicopter (CATIA v.4 model)

The volume grid generation started from the EC145 surface definition of Figure 6 already available in the design department, as a CAD CATIA model. The complete helicopter model has been cleaned and simplified by removing main and tail rotors, landing gear, antennas and handles. The air intakes and jet exhausts have been closed and all gaps between adjacent surface patches have been repaired [18]. Figure 7 shows the aerodynamic "water tight" CATIA model obtained after the above mentioned activity. The aerodynamic CATIA model has been then imported into the volume grid generator ICEM-Hexa. The resulting structured multi-block Navier-Stokes mesh about the isolated fuselage is composed of 64 blocks, 4.9 million nodes and allows 3 levels of multigrid. The topology structure is a C-O, with the C-structure in the longitudinal direction and the O-structure in the transversal one. A boundary layer grid has also been generated around the whole surfaces. Figure 8 shows the middle co-ordinate grid plane. The C-topology structure in the grid longitudinal direction is visible here.

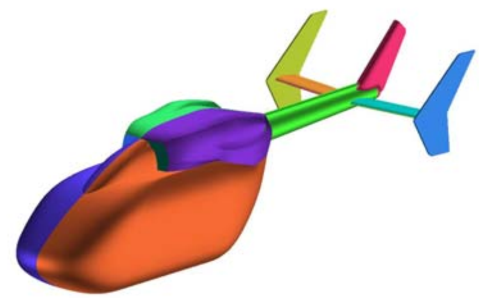

Figure 7 EC145 isolated fuselage (CATIA v.4 aerodynamic model)

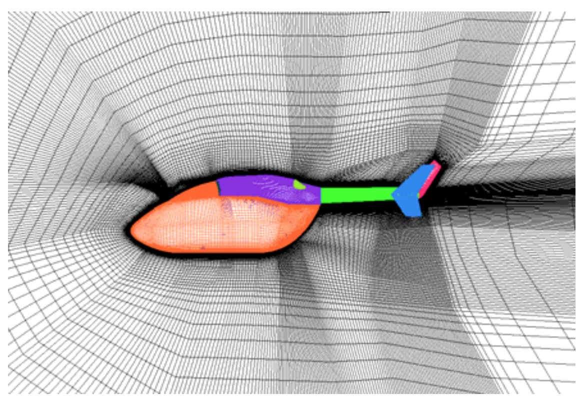

Figure 8 ICEM-Hexa mesh about the EC145 isolated fuselage: middle plane.

Figure 9 shows the pressure coefficient distribution on the EC145 helicopter fuselage at an advancing velocity of 40m/s and an angle of attack of α=0°. The aerodynamic department of Eurocopter is often asked to provide such kind of information - in limit flight conditions - to the static department. The static expert applies the aerodynamic loading to his FEM structural model to verify the resistance of components, such as windows, doors, etc.

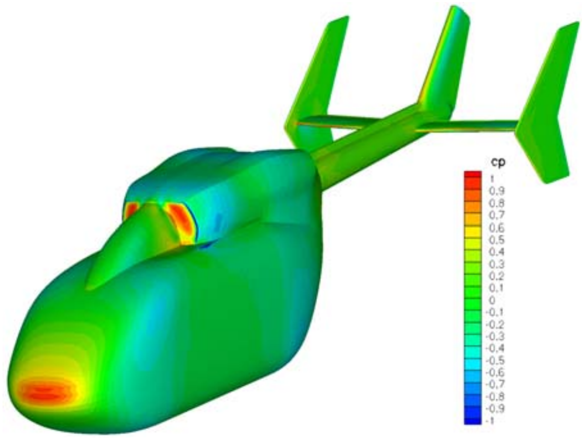

Figure 9 Pressure coefficient distribution on the isolated fuselage of the EC145 helicopter at α=0°.

Another useful information is the position and intensity of the wake vortices generated by the steep junction between the fuselage body and the tail boom. The separation taking place in this region gives birth to complex vortical structures which, convected downstream, might interact, depending on the flight condition, with the horizontal stabilizers, thus modifying drastically their aerodynamic response. Figure 10 shows the total pressure on transversal planes behind the fuselage in the same flight conditions of Figure 9. It can be noticed that the vortical structures indeed interact with the stabilizers. In order to correctly predict these interactional phenomena, good conservation properties of the numerical scheme and sophisticated URANS turbulence model, such as the 2-equation SST or LEA k-ω, are mandatory.

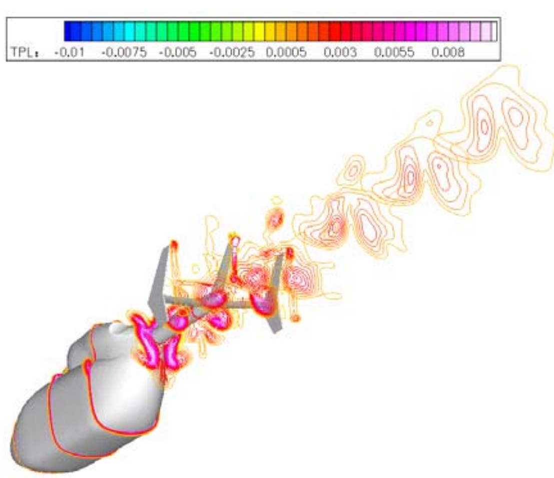

Figure 10 Total Pressure Losses on transversal planes behind the EC145 fuselage at α=0°.

Actuator Disk Modelling for the EC145 [20], [22]

In order to consider the effect of the main rotor without having to spend the large effort for a time-accurate simulation, quasi-steady simulations with so-called actuator disks are carried out. The main rotor is replaced by an infinitely thin disc, under which source terms are specified in order to mimic the influence of the rotor on the fluid [19]. This disc is practically accounted for as an inter-block cut boundary during the mesh generation process. Then a force distribution, provided for instance by a previous isolated rotor computation, can be interpolated as source terms on the surface mesh of the disc: the force itself in the momentum equation and its corresponding energy in the energy equation. As such, this formulation is readily amenable to low-velocity preconditioning and has been integrated in a parallel framework. For details the reader is referred to [20]. The loads applied on the AD surface have been derived from an already performed Navier-Stokes chimera computation about the isolated 4-bladed rotor, mounted on the EC145 helicopter, in forward flight conditions [21] (M=0.208, Re=4.5 Mill., μ=0.32).

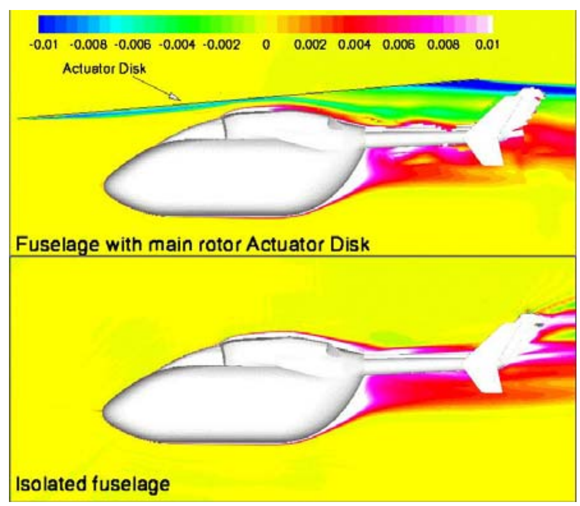

Figure 11 Total Pressure Losses distribution on the middle longitudinal fuselage section, with (upper) and without (lower) AD

The comparison between isolated fuselage and fuselage plus AD shows that the AD induced flow has a small effect on the total drag value (less than 4%), but an important one on the lift value (about 40%), highlighting that the AD model might be important in analysing where the rotor wake interacts with the fuselage components, for instance for pitch up or tail shake phenomena. This is evident when comparing the pressure distribution in the middle section of the horizontal stabilizer in Figure 12.

In a similar way the tail rotor can be modelled with an actuator disk [22]. The problem of properly setting up the grid system gets more complicated in this case. As a matter of fact, very little space was left between the various components: fuselage, main and tail disk. Masks had to be defined with care in order to make sure that the chimera hole boundaries lead to well defined interpolations. The final setting of masks is displayed in Figure 13.

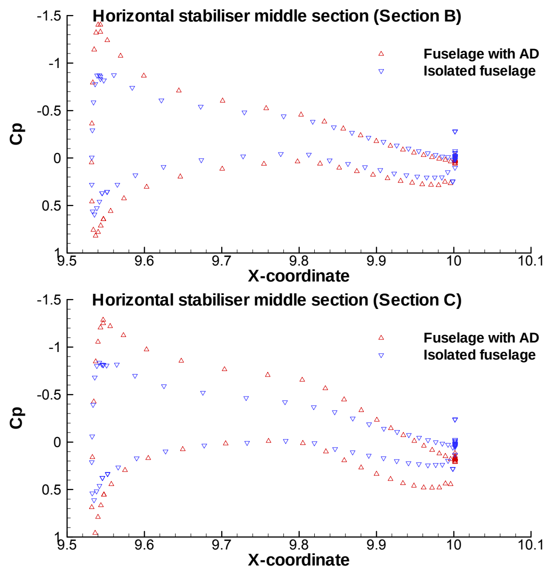

Figure 12 Pressure coefficient distribution on the middle section of the left (Section B) and right (Section C) horizontal stabiliser

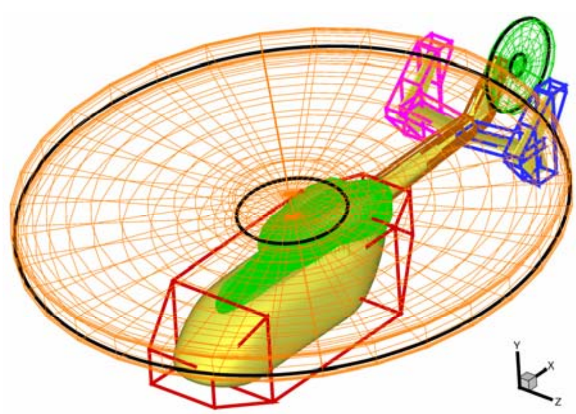

Figure 13 System of masks for the fuselage and the actuator disks

The global wake system is visualized in Figure 14 via the total pressure gains. Even in this quasi-steady simulation the complicated interaction between the wake system and the tail unit can be observed. It is hoped that such kind of numerical investigations will ease in the future the design of tail units. An extension of this work including engine in- and outlet and a specific numerical treatment for hot air flows is described in [23].

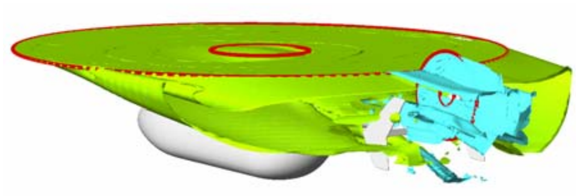

Figure 14 Total pressure gains for EC145 configuration with 2 actuator disks

Weak versus strong coupling on the 7A rotor [24]

As numerical methods reach a high level of maturity in the field of rotor aerodynamics, it is expected that significant progress in performance prediction of rotors can be achieved by including the structural response. The current state of the art consists in two different approaches for coupling dynamic and aerodynamic codes, namely weak and strong coupling.

Since weak coupling includes the rotor trim inherently, a trimmed solution for the strong coupling is a necessary prerequisite for a comparison of both coupling approaches and an automatic method was developed combining weak and strong coupling. In addition, the latter method has been compared with a manual trim.

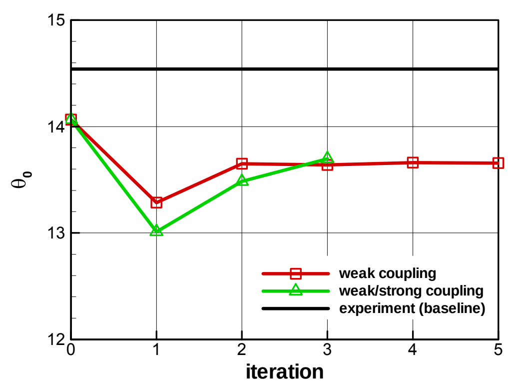

Figure 15 Trim convergence: collective pitch angle

The flight mechanics tool HOST is the computational environment delivering the blade dynamics to the coupled simulations, whereas CFD simulates the aerodynamics of the 7A model rotor. As far as the trim process is concerned when using the strong coupling approach, either an automatic trim (called weak/strong coupling) or a manual trim have been used. For this last case, a Jacobian matrix is computed with the simplified aerodynamic model of HOST in order to adjust the trim values to the desired ones.

The convergence of the weak coupling method is quite fast, as pointed out by previous studies [25], [26]. In the present case, five iterations were sufficient to obtain a very good stabilization of the control angles.

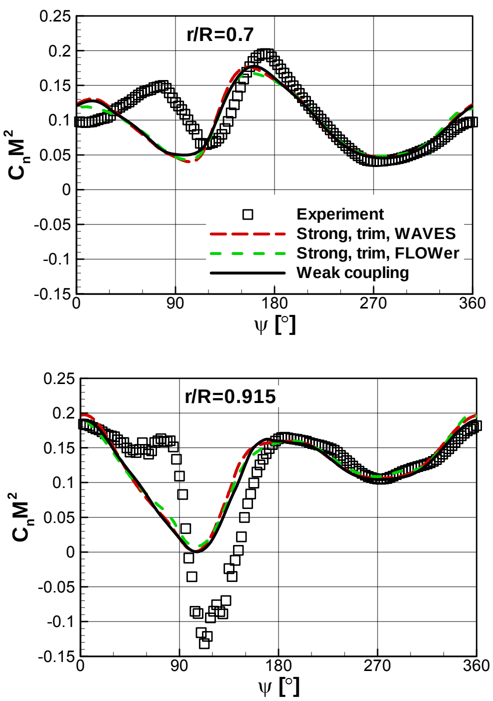

Figure 16 Comparison of Weak and. Strong Coupling calculations with experiment (lift)

Since the weak/strong coupling trim procedure is in principle based on the same trim methodology, the convergence of the collective pitch is included in Figure 15. The trim convergence is not quite as fast, i.e. the control angles do not stabilize in the same fast manner. However, it yields similar results in terms of control parameters. Here, the fact comes into play, that structural periodicity after a trim iteration is not forced by prescribing the deflections as harmonic series on the fluid side when converging the strong coupling but periodicity (of the entire aeroelastic system) develops itself only slowly and normally needs far more than 4 revolutions.

Weak and strong coupling (trimmed) are compared to experiment in terms of sectional lift coefficient (Figure 16) for inviscid computations. This plot shows that weak and strongly coupled calculations are very similar, and no significant improvements are brought by the quite complex trimmed strong coupling calculations.

As a consequence, if the strong coupling calculation is trimmed and converges to a periodic solution, the only difference between the weak and strong coupling results lies in the Fourier analysis performed on the corrections transmitted by the aerodynamic solver to the dynamic solver HOST. In the present case, this Fourier analysis is done on quantities stored at each degree, which means that the first 180 harmonics of the aerodynamic loads and moments are considered in the weak coupling, whereas no filtering is done in the strong coupling calculations. In this case, we can clearly understand that such a high filtering should have a negligible influence on the final results, since no excitation with a harmonic content higher than the 180th harmonic is generally encountered by the blades.

Interaction of 7A rotor with model support [27]

In order to validate the coupling of CFD with HOST dynamics for helicopter-like configurations (i.e. accounting for rotor-airframe interactions), the test case selected was the 7A rotor in interaction with its model support in the ONERA S1MA wind-tunnel (Figure 17). Indeed, this configuration combines the 7A rotor rotating above its model support, and the unsteady interaction between the rotor and the model support is similar to that between the main rotor and the fuselage of the helicopter.

Figure 17 View of the 7A rotor with model support

The interest of this selected configuration is the high-instrumentation of the 7A rotor, with 116 unsteady pressure transducers and 30 strain gauges for determining the blade deformation by strain pattern analysis (SPA), which allows a detailed validation of the aero-elastic computation. The details of the work were published in [27], and only some typical results are presented here.

The computation of bodies in relative motion and close proximity requires the use of specific numerical techniques with enough versatility to handle this kind of situation. The Chimera method, which is particularly well adapted to helicopter problems, was used in the present application. A background grid is attached to the model support (Figure 18), and split into 25 blocks with a total number of points of about 1.7 million. Among these, two main cylindrical blocks were defined for limiting the Chimera search and interpolation process. Individual child meshes are attached to each one of the four blades, and move freely inside the background grid. They are made of about 300,000 mesh points each. To account for blade flexibility effects, these child meshes can distort with time, the blade deformation being externally computed by HOST. For viscous computations, the mesh is clustered close to the walls in order to end up with y^+ values of the order of 1.

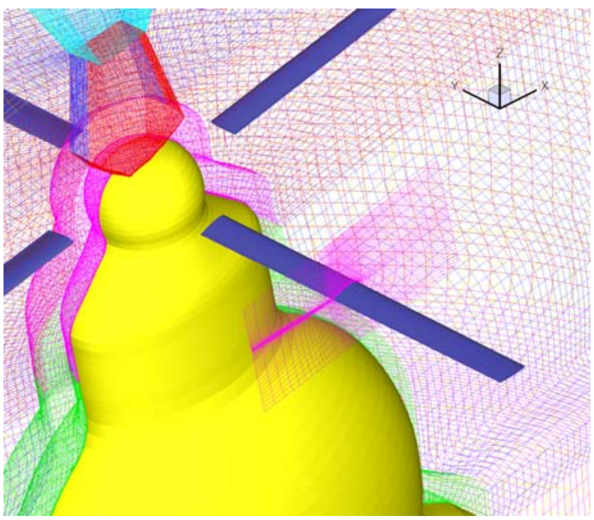

Figure 18 Close view of the mesh system

The unsteady time-integration is computed using a second order implicit Gear time-integration scheme with Newton sub-iterations in order to converge the time integration at each time step. The implicit system is solved with LU decomposition. The viscous computation is performed with fully turbulent flow using the k-ω model with SST correction and Zheng limiter in order to limit the dependency of the solution to the level of ω outside of the boundary layer. The iso-pressure contours at a given azimuth are plotted on the blades and the model support in Figure 19. The streamlines emitted upstream, in the plane of the rotor disk, clearly show the interaction of the rotor with the model support.

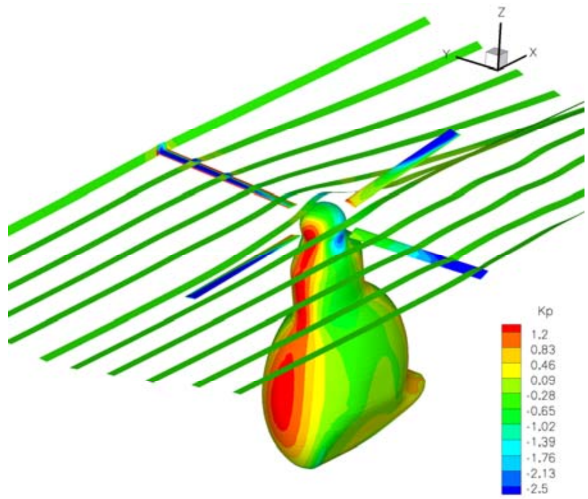

Figure 19 Iso-pressure contours and streamline pattern around the rotor with model support

Close to the root of the forward blade, the support induces an upwash velocity which tends to increase the local incidence, while the rear blade is embedded in the downwash induced by the model support. This downwash extends over a large part of the blade span, reducing the local incidence. This is clearly noticeable when looking at the lift evolution computed with and without model support (Figure 20). Here, only the most inboard pressure-instrumented blade section is plotted, comparing experiment with the 2 sets of computations. The large influence of the model support is quite clear, especially for the rear blade around 0º azimuth, but also for the forward blade around 180º azimuth. It should also be noted that the computation with model supports correlate better with the experimental evolution of lift. The previous computation was performed with trim conditions and blade dynamics provided by HOST and used as a prescribed boundary condition by the CFD. In that case, the blade dynamics and trim are obtained by solving the fluid-structure coupling problem with the simplified aerodynamics model of HOST. A step further in the computation is obtained when using the weak coupling between HOST and CFD, where an iterative solution is obtained, rotor revolution after rotor revolution, between HOST and the CFD, the HOST computation prescribing a CFD correction to the simplified aerodynamics in order to end up with a consistent solution between HOST and CFD.

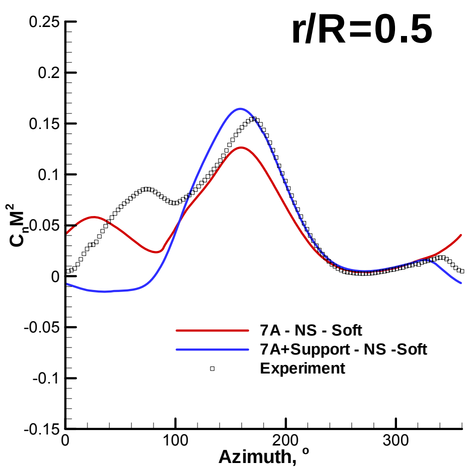

Figure 20 Comparison of lift evolution at mid-span of the blade with and without model support

Such an iterative coupling converges quite rapidly as shown in Figure 21 for the longitudinal pitch angle. Similar convergence can be observed for the other rotor control parameters. Not surprisingly, this parameter is the most affected one by the model support, and a difference of 1.5° in longitudinal pitch angle is obtained between the computation without and with model support. Furthermore, the solution with model support provides a better prediction of this longitudinal pitch angle when comparing with the S1MA data.

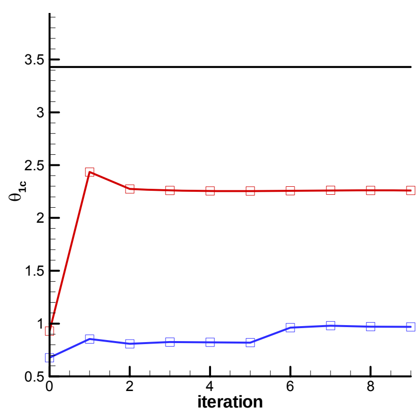

Figure 21 Convergence of longitudinal pitch angle with (red) and without (blue) model support

Dauphin 365N main rotor-fuselage configuration [28]

The time-accurate simulation of a main rotor-fuselage configuration requires the combination of the numerical elements which were described in chapter “DGV fuselage” and “Interaction of 7A rotor with model support”, i.e. grids of high quality around the fuselage and the rotor blades, a RANS solver with appropriate turbulence models and a chimera method well adapted to rotorcraft problems. Here the elsA solver was applied with the k-ω SST turbulence model on a grid with a total of 2.6 Mill. grid points in order to prove the feasibility of such a flow simulation. The test case chosen is a low speed test case (M_∞=0.044, μ=0.15) with a strong main rotor-tail unit interaction. The experimental data was obtained in the S2-Chalais-Meudon and F1-Fauga-Mauzac wind tunnel on a Dauphin 365N model. The blade motion was simplified as a rigid body motion in order to reduce the computational effort and the complexity to run such a computation. This simplification is acceptable in the context of a feasibility study. A detailed discussion of the flow simulation and a comparison of the computed with experimental data is given in [28]. As an example Figure 22 presents the vorticity in a section parallel to the free stream direction showing the interaction of a rotor blade with the wake system. The surface colours present the pressure coefficient.

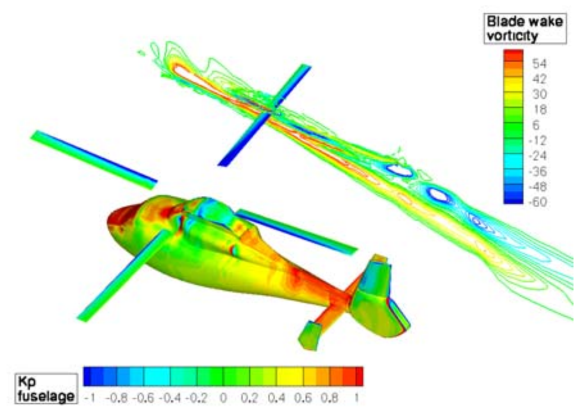

Figure 22 Pressure coefficient on the fuselage surface and vorticity in a section for the Dauphin 365N model in low speed flight

Complete BO105 Wind Tunnel Model [32]

As the last example a very complex configuration is considered, i.e. the BO105 wind tunnel model (1:2.5 scale) consisting of the fuselage with horizontal and vertical stabilizers, skids, spoiler, wind tunnel support strut, the main and tail rotor. The FLOWer computation reported here simulates a forward flight test case measured in the HeliNOVI test campaign [29], [30] in the DNW LLF. The flight conditions are M_∞=0.177, $M_{(\omega R)MR}$=0.65, $M_{(\omega R)TR}$=0.63, α=-5.2°.

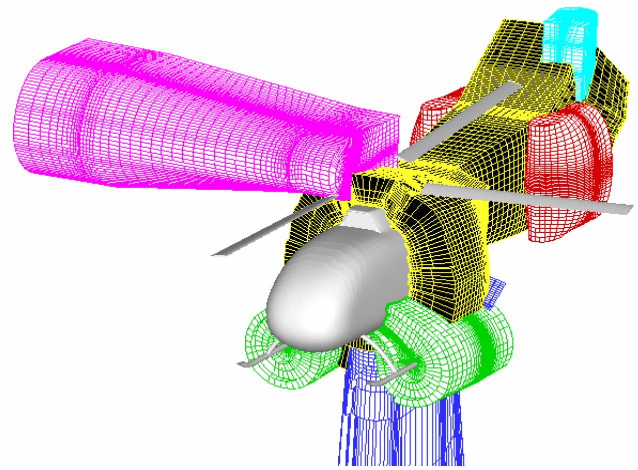

Figure 23 Near-field grids around the BO105 wind tunnel model

The chimera technique was used for enabling the relative motion of the blades and the fuselage components and for reducing the grid generation effort. For each component a grid (near field grid) was generated separately and the whole configuration was built by connecting the grids via chimera functionalities, Figure 23. For further details on the chimera technique please refer to [31]. The corresponding background grid, Figure 24, was created with a mesh generator for Cartesian meshes which are automatically adapted to the near field grids with a grid coarsening from the body surfaces to the far field enabled by the use of discontinuous cut boundaries [31].

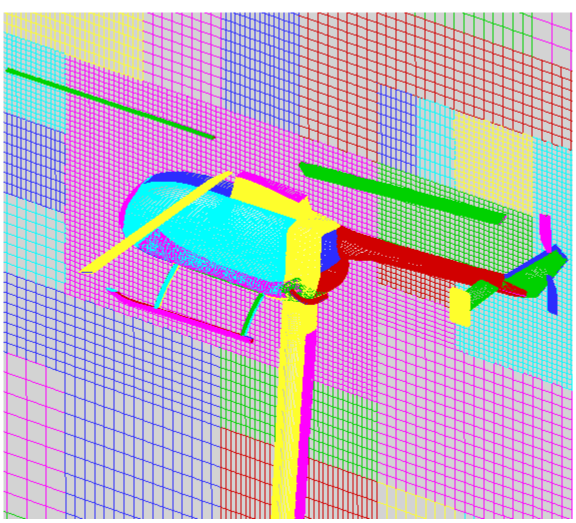

Figure 24 Surface grid and a cross section through the Cartesian background grid of the chimera grid system for the BO105 model

A slightly modified version of the Wilcox k-ω turbulence model was applied. Figure 25 presents the instantaneous pressure distribution in the mid-section on the surface of the fuselage for ψ_{MR}=0. Further details on this RANS simulation are found in [32].

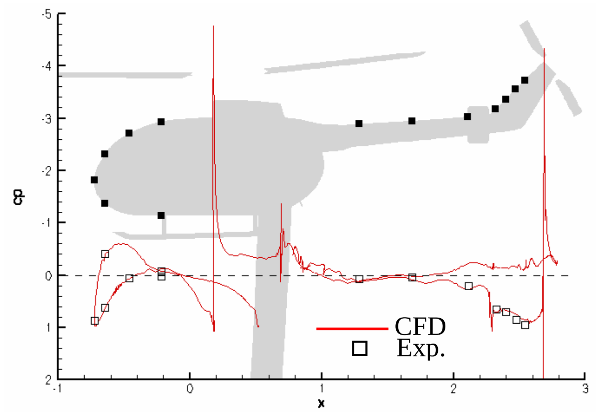

Figure 25 Instantaneous pressure distribution in the mid-section on the fuselage surface (ψMR=0)

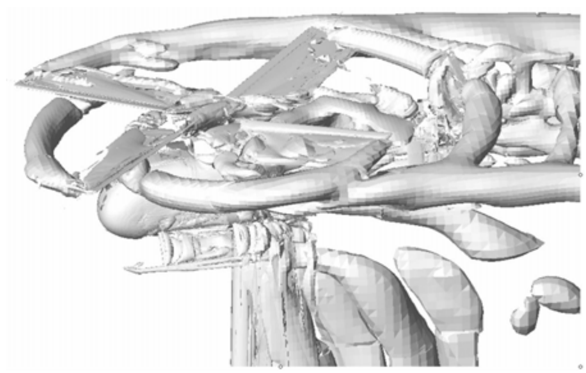

Figure 26 Iso-vorticity surfaces

The very complex wake system of such a rotorcraft flow is presented in Figure 26 by iso-vorticity surfaces. Although such a figure gives only a qualitative presentation it shows that there are many more sources for wake generation than just the rotor blades. In fact it can be observed that the engine cowling, the fuselage back and the skids produce wakes which contribute to the wake system and create a highly unsteady interactional flow.

Conclusions

The work presented in this paper allowed giving an overview of the work completed both in France and Germany during the CHANCE project. The main initial objectives were reached. All the basic functionalities necessary for computing complete rotorcraft configurations were developed step by step, validated and integrated into a general multi-purpose software tool. Code delivery and integration into the industrial environment was regularly completed all along the project. Intermediate steps related to isolated components as well as simplified models of the helicopter were also considered in order to get products immediately usable for the applications. Furthermore, the work also included the account of blade elasticity effects in the aerodynamic simulation by coupling the CFD solutions with helicopter comprehensive analysis, using either a weak coupling (rotor revolution per rotor revolution) or a strong coupling (azimuthal step per azimuthal step) procedure.

The present project thus allowed providing the basic tools which are necessary for computing complete rotorcraft configurations and which can be used by industry for design and analysis of aerodynamic problems. Furthermore, these tools form the basis for new studies more focused on the specific difficulties of helicopters, such as the fine description of interactional phenomena and of dynamic stall.

Acknowledgments

The authors are willing to acknowledge the French and German Official Services (DPAC, DGA and BMWA) for funding the CHANCE project. Furthermore, they are also very thankful to all their numerous colleagues at DLR, ONERA, EC, ECD and IAG who very actively participated in this 6-years cooperative program.

References

[1] Meakin, R.L., Wissink, A.M., "Unsteady aerodynamic simulation of static and moving bodies using scalable computers", AIAA Paper 99-3302, 1999.

[2] Hariharan, N., Sankar, L.N., "Higher order numerical simulation of rotor flow field", American Helicopter Society 49th Forum and Technology Display, May 1994.

[3] Tang, L., Baeder, J.D., "Accurate Euler simulation of parallel blade-vortex interaction", American Helicopter Society 53rd Annual Forum, April 1997.

[4] Benoit, C., "Numerical simulation of 2D blade vortex interaction using moving overset grids", 25th European Rotorcraft Forum, September 1999.

[5] Ochi, A., Aoyama, T., Saito, S., Shima, E., Yamakawa, E., "BVI noise predictions by moving overlapped grid method", American Helicopter Society 55th Annual Forum, May 1999.

[6] Ahmad, J.U., Strawn, R.C., "Hovering rotor and wake calculations with an overset-grid Navier-Stokes solver", American Helicopter Society 55th Annual Forum, May 1999.

[7] Wenren, Y., Fan, M., Wang, L., Xiao, M., Steinhoff, F., "Application of Vorticity Confinement to prediction of the Flow over Complex Bodies", AIAA Journal, Vol. 41, No 5, May 2003.

[8] Sidès, J., Pahlke, K., Costes, M., "Numerical simulation of flows around helicopters at DLR and ONERA", Aerospace Science and Technology, Vol. 5, pp 35-53, 2001.

[9] E. Schöll, "Numerical simulation of the BK117 / EC145 Fuselage Flow Field.", Proceedings of the 25th European Rotorcraft Forum, Rome, Italy, September 1999.

[10] Gazaix, M., Jollès, A., Lazareff, M., « The elsA object-oriented computational tool for industrial application », 23rd ICAS Conference, September 2002.

[11] Kroll N., Rossow C.C., Becker K., Thiele F., MEGAFLOW-A Numerical Flow Simulation system, ICAS-Congress, Melbourne, Australia, Sept. 1998.

[12] Kroll, N., Rossow, C.-C., Becker, K., Thiele, F., "The MEGAFLOW project", Aerospace Science and Technology, Vol. 4, June 2000.

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

[14] van der Wall B. G., "Analytic Formulation of Unsteady Profile Aerodynamics and its Application to Simulation of Rotors", DLR-FB 90-28, 1990.

[15] Costes, M., Gavériaux, R., Renaud, T., "Application of CFD to the computation of complex rotorcraft configurations", ECCOMAS 2004, Jyväskylä, July 2004.

[16] Khier, W, "Application of Cartesian Background Grid in Combination with Chimera Method to predict the Aerodynamics of Helicopter Fuselage", 29th European Rotorcraft Forum, September 2003.

[17] Splettstoesser, W. R., Junker, B., Schultz, K.-J., Wagner, W., Weitemeyer, W., Protopsaltis, A., Fertis, D., "The HELINOISE Aeroacoustic Rotor Test in the DNW - Test Documentation and Representative Results, DLR-Mitteilung 93-09.

[18] D´Alascio, A., Pahlke, K., Le Chuiton, F., "Application of a Structured and an Unstructured CFD-method to the Fuselage Aerodynamics of the EC145 Helicopter. Prediction of the Time Averaged Influence of the Main Rotor.", European Congress on Computational Methods in Applied Sciences and Engineering ECCOMAS 2004, Jyväskylä, July 2004.

[19] Le Chuiton, F., "*Actuator Disc Modelling for Helicopter Rotors",* Aerospace Science and Technology, Vol. 8, pp 285-297, 2004.

[20] Le Chuiton, F., "*Actuator Disc Modelling for Heli-*

copter Rotors", Proceedings of the 28th European Rotorcraft Forum, Bristol, UK, September 2002.

[21] D'Alascio, A., Castellin, C., Costes, M. and Pahlke, K., "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.

[22] Le Chuiton, F., "Progress Towards the Simulation of the Complete Helicopter with Actuator Disks", STAB Symposium 2004, Bremen, Germany.

[23] Le Chuiton, F., "*Quasi-Steady Simulation of a Complete EC-145 Helicopter Fuselage + Main/Tail Actuator Disc + Simulated Engines*", Proceedings of the 31st European Rotorcraft Forum, Florence, Italy, September 2005.

[24] Altmikus, A. Wagner, S., Beaumier, P., Servera, G., „ A Comparison: Weak versus Strong Modular Coupling for Trimmed Aeroelastic Rotor Simulations.", 58th AHS Annual Forum, June 2002.

[25] Servera, G., Beaumier, P., Costes, M.: "A weak coupling method between the dynamics code HOST and the 3D unsteady Euler code WAVES", 26th European Rotorcraft Forum, The Hague (The Netherlands), Sept. 2000.

[26] Pahlke, K. and van der Wall, B., "Chimera simulations of multibladed rotors in high-speed forward flight with weak fluid-structure-coupling", AST, Volume 9, Issue 5, Pages 379-389, 2005.

[27] Rodriguez, B., Benoit, C., Gardarein, P., "Unsteady computations of the flowfield around a helicopter rotor with model support", 43rd AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, January 2005.

[28] Renaud, T., Le Pape, A., Benoit, C. "Unsteady Euler and Navier-Stokes computations of a complete helicopter", 31st ERF, Florence, September 2005.

[29] Langer, H.-J., Junker, B., Plassmeier, V., Buchner, F., Mikulla, V., Mercker, E., "The Unique Capabilities of a Complete Mach-Scaled Helicopter Model for the DNW-LLF", 27th European Rotorcraft Forum, Sept.11-14, 2001; Moscow, Russia.

[30] Yin, J., van der Wall, B., Oerlemans, S. et al., Representative Test results from HeliNOVI Aeroacoustic Main Rotor/Tail Rotor/Fuselage Test in DNW. 31st European Rotorcraft Forum, 13-15 September 2005, Florence, Italy.

[31] Schwarz, T., "The Overlapping Grid Technique for the Time-accurate Simulation of Rotorcraft Flows", 31st ERF, Florence , September 2005.

[32] Khier, W., Schwarz, T. "Time-accurate simulation of the flow around the complete BO105 wind tunnel model", 31st ERF, Florence, September 2005.