

INDUSTRIAL VALIDATION OF NUMERICAL AERODYNAMICS ABOUT ROTOR HEADS: TOWARDS A DESIGN OPTIMISATION AT EUROCOPTER

F. Le Chuiton, T. Kneisch, S. Schneider and Ph. Krämer
Eurocopter Deutschland GmbH, Munich, Germany

Abstract

In view of reducing the overall drag of the helicopter, activities at Eurocopter Deutschland have been started in order to be able to investigate and minimise the rotor head contribution. Besides traditional wind tunnel and flight tests, two approaches have been newly applied to the aerodynamics of the rotor head: CAD surface based Computational Fluid Dynamics computations (FLOWer and TAU flow solvers of DLR) and an analytical geometric modelling. The CFD computations provide an in-depth insight in surface pressure distribution, surface flow features and unsteady three-dimensional vortex structures of the wake behind the rotor head. Moreover, global loads as well as a detailed loads breakdown over components are of value for design activities. The analytical geometric modelling makes use of a subdivision of the rotor head into elemental shapes, for which loads are separately computed and then integrated over the whole geometry. The model is then substantiated with CFD results and also with data from wind tunnel and flight tests.

1. INTRODUCTION

In the context of ever stronger requirements for economical and ecological air transportation, the helicopter industry is urged among other things on reducing the fuel consumption of their machines. A key activity in this area is the shape optimisation of the fuselage in view of decreased drag values. In this respect, it is important not only to tackle the cell itself, cockpit and backdoor, but also to address components such as the landing skids and the rotor head, which make up a substantial part of the total drag.

As to the rotor head, the Eurocopter Group (ECG) participates to a number of national (SHANEL-L and ECO-HC in Germany and SHANEL in France) and European (Clean-Sky GRC2) research projects well supporting and gearing into company internal activities.

While Eurocopter SAS (EC) has recently focused on the study of the contribution of the rotor head wake to the aerodynamic behaviour of the helicopter ([4]), Eurocopter Deutschland (ECD) has begun putting efforts on the setting up of a process chain for optimised designs of rotor heads. Describing the present status of the latter is the subject of this paper.

So far, in order to deal with development issues (e.g. loads, performance) while focusing on the effects of the rotor head, ECD has relied on experimental data from wind tunnel or flight tests and analytical modelling. The added use on the one hand of an

extended analytical method and on the other hand of CAD surface based methods aims now at widening the range of treatment possibilities.

The new analytical model has been implemented into in-house global helicopter tools used at ECG, while the surface based tools are both Computational Fluid Dynamics (CFD) flow solvers developed by the German Aerospace Center (DLR).

Which tool is actually being used to answer a specific request is mainly a matter of compromising between criteria such as computing time, fast prototyping, fidelity of the physical modelling.

2. WIND TUNNEL EXPERIMENTAL DATA

A wind tunnel measurement campaign has been carried out at the University of Munich in 2008 on a wind tunnel adapted model at scale 1:7.333 (Figure 1) of the EC145 configuration. A simplified rotor head of an advanced 4-bladed rotor (Figure 2) was mounted and able to rotate with a prescribed collective pitch of 4° , however no cyclic controls could be used.



Figure 1: EC145 wind tunnel model

For this investigation the wind tunnel A of the University of Munich has been used in open test section mode, the characteristics of which can be seen online [1].



Figure 2: simplified 4-bladed rotor head model

Besides the measurement of surface pressures and flow velocities on PIV windows in the wake of the backdoor, a series of loads polars in incidence and side-slip along with the rotor-head mounted or not have been measured. For further details, please, refer to [13] and references therein.

In order to derive polars for the isolated rotor head, the subtraction method has been used, in that data on the one hand of the isolated fuselage and on the other hand of the configuration with rotating rotor head have been subtracted from each other. The fact that the incremental loads of the rotor head are relatively small compared to those of the full configuration may explain the relative scattering of results (dots in Figure 20 of section 4.3.2).

The CFD study presented below aimed not only at the validation of a simulation with a rotating rotor head but also at studying the effect of its wake impinging on the empennage, which however is not part of the scope of the present paper. Hence in order to account for both CFD objectives a fuselage incidence of 10° has been retained, so as to maximise the interaction with tail surfaces.

Table 1: wind tunnel conditions

$V_\infty =$	40	m/s
$(\alpha, \beta)_{\text{FUSE}} =$	(10 , 0)	deg
$\alpha_{\text{SHAFT}\% \text{FUSE}} =$	-5	deg
$T_\infty =$	24	$^\circ\text{C}$
$\rho_\infty =$	1.119	kg/m^3
$\omega =$	169.3	rad/s
$M_\infty =$	0.12	---
$\text{Re}_\infty =$	2.4×10^6	m^{-1}
$\mu =$	0.315	---

These conditions have then been simulated by all numerical methods and checked against wind tunnel data.

3. THE CFD APPROACH

Both flow solvers developed at DLR: FLOWer following a multi-block structured approach and TAU implemented for the unstructured one are in use at ECD. For a general presentation of the codes, please refer for instance to [6] and [12] as regards FLOWer and to [5] and [11] concerning the TAU system.

Since the process of migrating from the first to the second one is still ongoing, both have been used in the present study.

3.1. Code Description

The FLOWer code solves the compressible three-dimensional unsteady Reynolds-averaged Navier-Stokes equations (URANS) on structured multi-block grids. Helicopter applications can be run through a fully general motion module and the use of a non-hierarchical implementation of the Chimera technique. Both allow accounting for complex grid systems with overlapping sub-grids in relative motion to each other. The spatial discretisation, based on a cell-centred finite volume formulation, makes use of central differences augmented by a scalar artificial dissipation operator. The time discretisation is implemented with backward Euler differences and integration is carried out through a 5-stage Runge-Kutta scheme. Convergence is accelerated using local time stepping, implicit residual smoothing and 3-level V-multigrid cycling. Turbulence effects can be accounted for by a series of algebraic, 1 or 2-transport equation or even Reynolds-stress transport (RSM) models.

The TAU system solves the URANS equations on unstructured grids and supports all four primary element types: tetrahedra, hexahedra, prisms and pyramids. The spatial discretisation is based on a cell-vertex finite volume formulation thereby using the dual grid, which is computed during a pre-processing step. Central differences along with scalar artificial dissipation have also been used for the convective fluxes and low velocities are accounted for by a pre-conditioning technique. TAU offers a wide range of turbulence models, ranging from simple algebraic ones, 1- and 2-transport equation models to full RSM models. All TAU simulations described in this paper made use of the 2-transport equation Menter-SST model. The convective fluxes of turbulence equations are discretised with the AUSMDV scheme. The time derivative is also discretised using implicit backward Euler differences and the discrete equation system is integrated by an implicit LU scheme. Convergence acceleration is implemented by local time stepping, residual smoothing and a multigrid method. Here also, the Chimera technique in combination with a motion module enables the

simulation of complex overlapping and moving grid systems. Finally, greater flexibility for complex simulations is ensured through wrapping Python classes that allow direct addressing of the main subroutines.

3.2. Isolated Advanced 5-Bladed Rotor Head

In a preliminary stage the global loads of an isolated full-scale rotor head have been investigated using both flow solvers. The configuration refers to a high speed forward flight, the conditions of which are listed in Table 2.

Table 2: isolated rotor head; flight conditions

$V_{\infty} =$	70.0	m/s
$\alpha_q =$	-3.7	deg
$\omega =$	39.4	rad/s
$T_{\infty} =$	279.5	K
$Re_{\infty} =$	4.2×10^6	m^{-1}

Figure 3 shows the simplified 5-bladed geometry of the rotor head. Due to the fact that the structured mesh generation implies a higher effort, the geometry retained for FLOWer was further simplified, in that the connection of the blades to the rotor mast has been removed. Figure 4 illustrates a cut through the volume mesh.

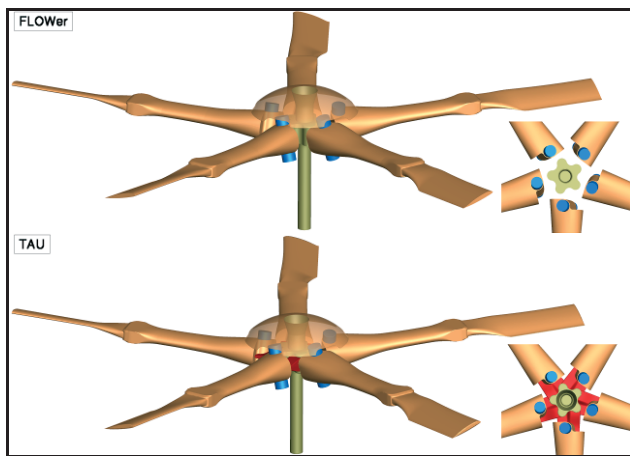


Figure 3: advanced 5-bladed rotor head; geometry

Table 3 summarises the statistics of the structured and unstructured grid systems.

Table 3: both grid systems

grid	blocks	nodes
structured	280	10.990.435
unstructured	1	7.384.059

First results of the structured approach with FLOWer are available. The distribution of the pressure coefficient (c_p) is illustrated in Figure 5 and the drag breakdown over several components after 10 rotor revolutions is shown in Figure 6. Since the

computation using the TAU code is still ongoing, the presentation of results is postponed to a future paper.

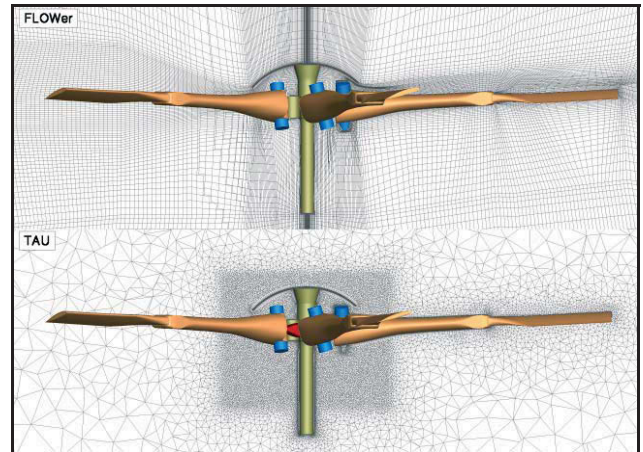


Figure 4: grids for the isolated rotor head

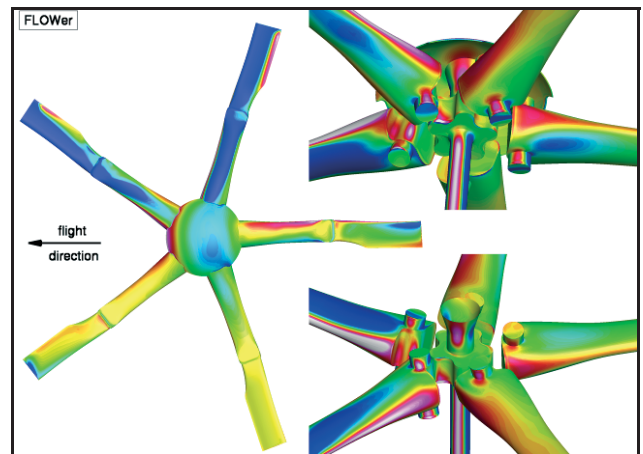


Figure 5: c_p -distribution (FLOWer)

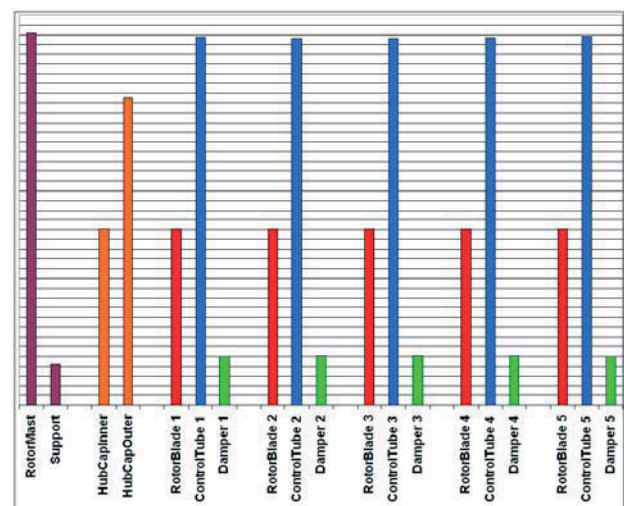


Figure 6: drag breakdown (FLOWer)

3.3. EC145 with 4-Bladed Rotor Head: Grids

The first grid system has been constructed with the Hexa module of the commercial grid generator ICEMCFD and is based on a structured multi-block

approach, in which seven sub-grids communicate with each other through Chimera interpolations. They accommodate: the helicopter fuselage (Figure 7) down to the far-field; the rotor head (Figure 8); the four pitch rods (Figure 9); and a fixed and heavily refined wake transport grid (marked in red in Figure 7). The latter is not mandatory within the described grid system, but has been used to improve the conservation of the vortex system generated by the rotor head. An iterative and effort intensive grid generation process eventually led to a satisfying grid quality and made it possible to reduce the number of orphan points to a maximum of two over a complete rotor head revolution. The key data of the overall grid system is presented in Table 4.

Table 4: overview of the structured grid system

grid	blocks	nodes
fuselage	196	11.990.000
rotor head	176	10.250.000
pitch rod	4 x 10	4 x 272.000
wake transport	8	1.920.000
total	420	25.238.000

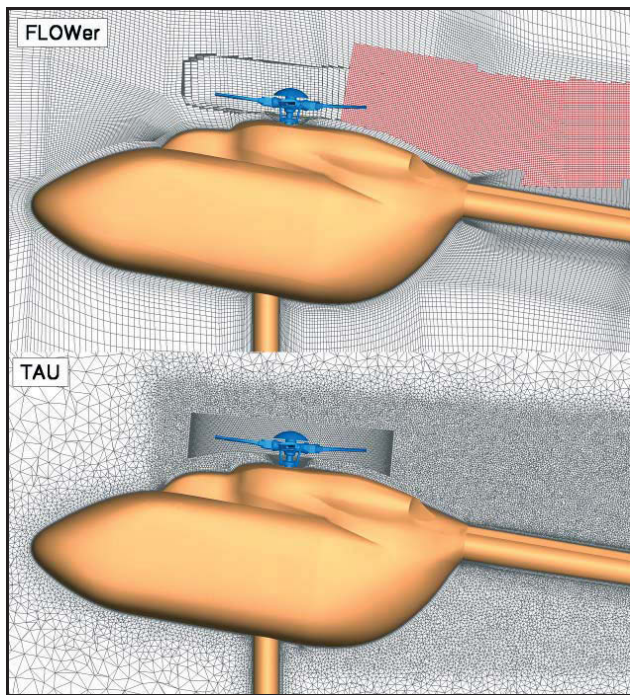


Figure 7: grids of the BK117-C2 fuselage

The second grid system has been prepared with the commercial grid generator Centaur and defines an unstructured grid consisting of two Chimera sub-grids. The background grid housing the fuselage down to the far-field along with a pre-defined hole is shown in Figure 7. For a better capturing of the turbulent wake behind the rotor head the grid has been refined by placing geometric sources. While the fuselage surface is mainly discretised with triangles and prisms in the boundary layer, the rotor head surface also consists of structured and

unstructured quadrilaterals and hexahedra in the boundary layer. Figure 8 illustrates the Chimera child grid of the rotor head. The key data is summarised in Table 5.

Table 5: overview of the unstructured grid system

grid	blocks	nodes
fuselage	1	7.600.000
rotor head	1	9.900.000
total	2	17.500.000

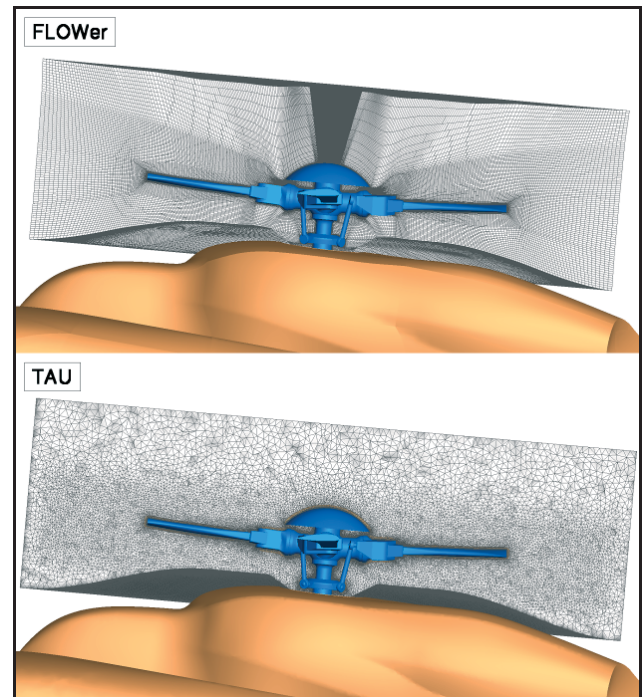


Figure 8: grids of the ATR-A rotor head

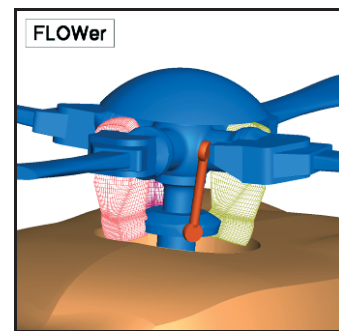


Figure 9: grid of the pitch rods (only FLOWer)

3.4. EC145 with 4-Bladed Rotor Head: Results

As already mentioned in the case of the isolated rotor head, TAU computations are still running at the time of writing this paper. Hence only FLOWer results are shown in the following, namely at the end of the unsteady run (5 revolutions), when the three-dimensional solution was saved.

In Figure 10, the c_p -distribution is shown. The rotation leads to an increased dynamic pressure on the advancing blade stub and conversely on the retreating one. Also the hub cap experiences under-pressure on its upper surface and over-pressure (not shown here) on the lower one, which brings up the lift component of the aerodynamic force. Additionally, the central parts: swash plate, mast and blade attachments are subject to a permanent ram drag.

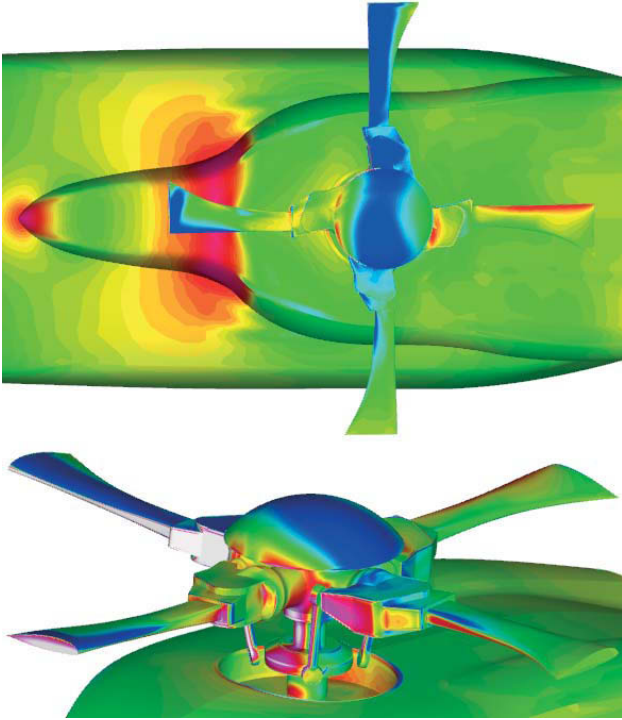


Figure 10: c_p -distribution (FLOWer)

Friction lines can be seen on Figure 11, where the interaction of the wake with the engine cowlings is evidenced by the flow separation along approximately one stub radius downstream. It also appears that the retreating blade stub is blown from the trailing edge, since it is located in the recirculation area.

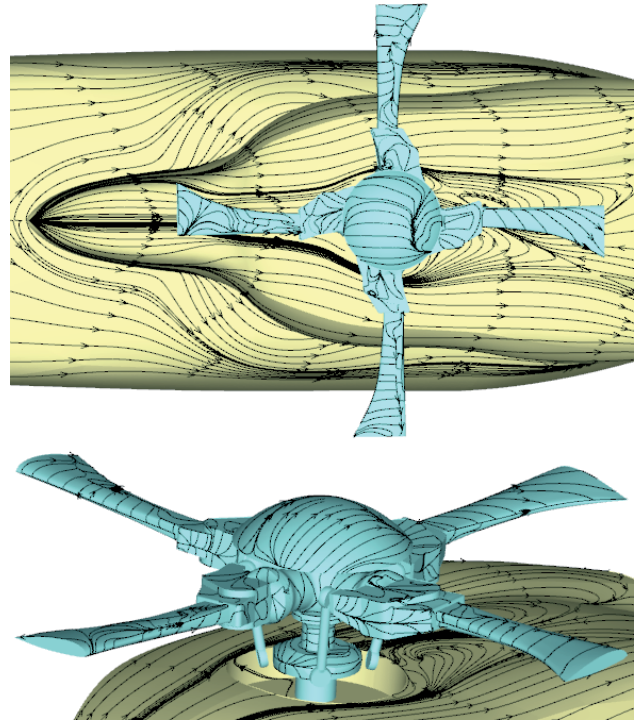


Figure 11: friction lines (FLOWer)

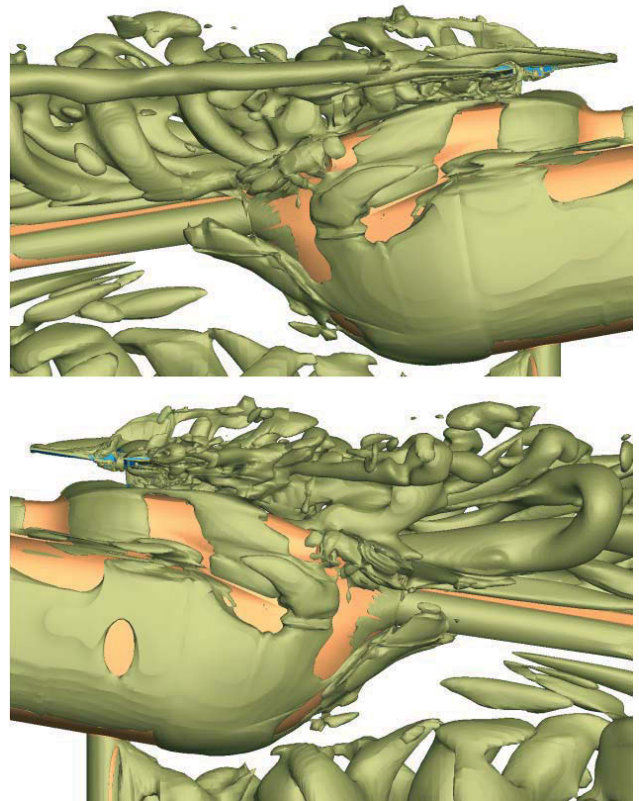


Figure 12: side-views of the N_k wake (FLOWer)

$$(1) \quad N_k = \sqrt{\overline{\overline{\Omega}} : \overline{\overline{\Omega}} / \overline{\overline{S}} : \overline{\overline{S}}}$$

where $\overline{\overline{S}}$ and $\overline{\overline{\Omega}}$ are the symmetric and skew-symmetric parts of the velocity gradient tensor.

The blade stub tip vortices as well as the vortices generated by the inner structure of the rotor head and hub cap are visible in the flow field downstream the rotor head.

3.5. Validation

The following figures show a comparison of both CFD computations presented in the previous section and of another non reported FLOWer run without

Cartesian block (shown in red in Figure 7) with the wind tunnel results. It is here reminded that TAU values are to be considered as temporary.

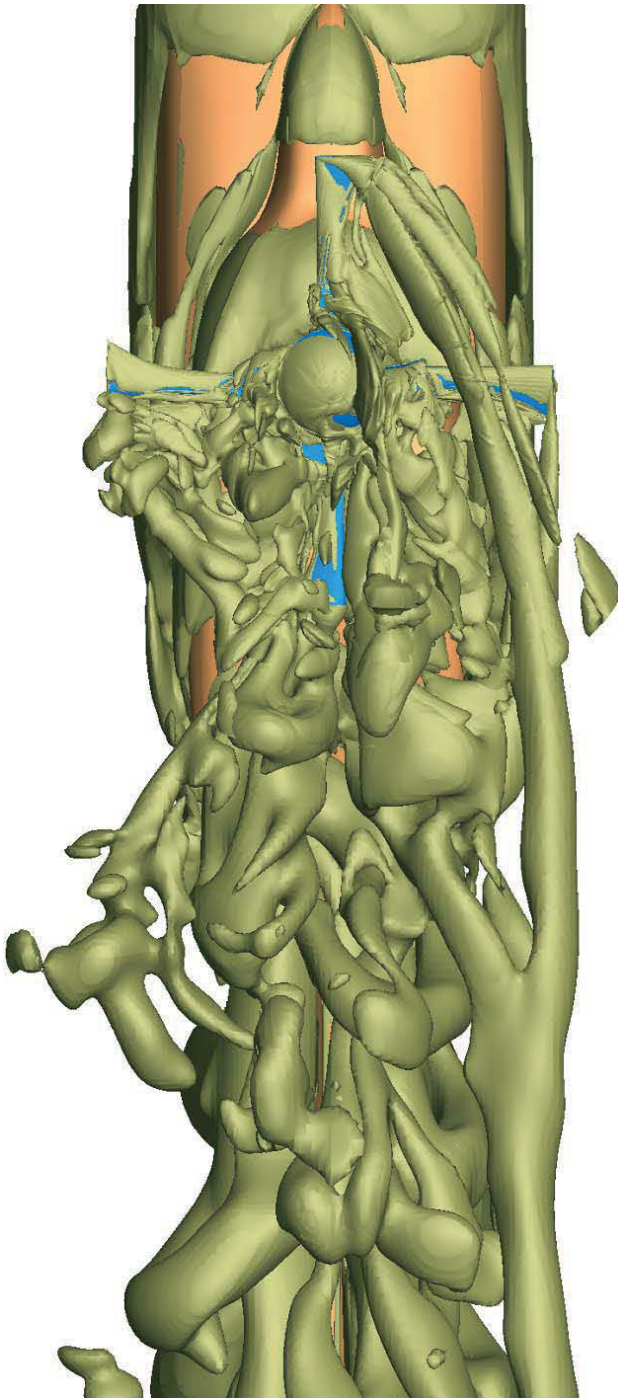


Figure 13: top view of the N_k wake (FLOWer)

Global forces are displayed in Figure 14. In accordance with previous observations, the drag appears to be overestimated in both FLOWer and TAU results, which is likely attributable to the difficulty of correctly capturing the massive flow separation behind the backdoor using the RANS approach. More advanced and expensive turbulence modelling, such as DES like methods or the recently introduced Scale Adaptive Simulation (SAS)

modelling in a URANS context, would probably allow getting closer to the experimental value.

Larger differences in lift are to be noticed too, which can be traced back to the particular incidence of the fuselage causing the horizontal stabiliser to lay a good portion more in the backdoor wake as at zero incidence. As it appears from a detailed load breakdown, the horizontal stabilisers are major contributors to the lift. Hence, the numerical capturing of the backdoor wake has an influence on the local flow conditions of the stabilisers and consequently on their effectiveness. A more properly computed wake would yield most probably a better comparison not only of the drag but also of the lift.

Despite the former two not yet optimal comparisons, the side force seems to be nicely hit. This is related to the fact that the vertical tail is virtually the only net contributor to the side force. Further, since it consists mostly of profiled elements, the RANS flow solvers used here perform much better than on blunt parts and thus can better approximate the wind tunnel experiment.

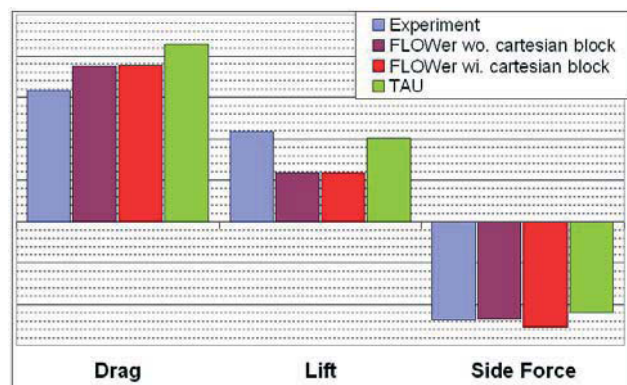


Figure 14: fuselage + rotor head; global forces

The global moments, in Figure 15, exhibit a much better comparison with experimental values, which also correlates with previous numerical experience. That the yaw moment be better captured, when expressed in percentage, than the pitch moment emphasizes the reasoning developed above for the forces.

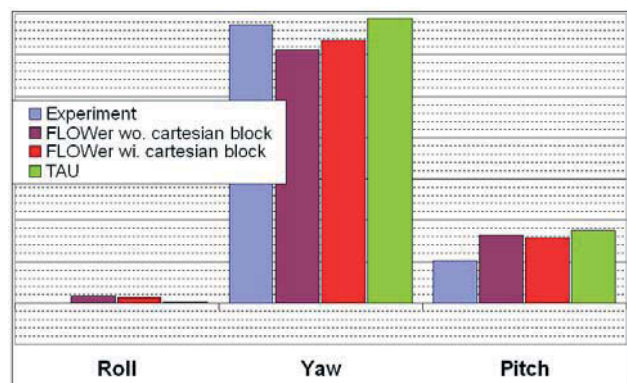


Figure 15: fuselage + rotor head; global moments

The global drag of the 4-bladed rotor head is shown in Figure 16, where both FLOWer and TAU exhibit a higher drag value than the wind tunnel data. However the FLOWer computations lie within 7.5% of the experimental value, which however lies within the confidence interval of the experimental results (see Figure 20).

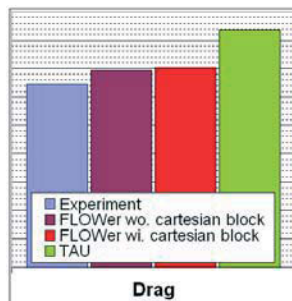


Figure 16: rotor head only; drag

The breakdown of the drag over the main rotor head components is shown in Figure 17. Both flow solvers yield very similar and encouraging results, though not for the mast and blade number 4. As to the latter, it is most likely attributable to the fact that the unstructured grid generation process could not ensure a rotational symmetric distribution of grid nodes. This was however possible using the structured approach, which results in identical FLOWer drag values over all four blade stubs. Hence the same should be aimed for in the unstructured approach.

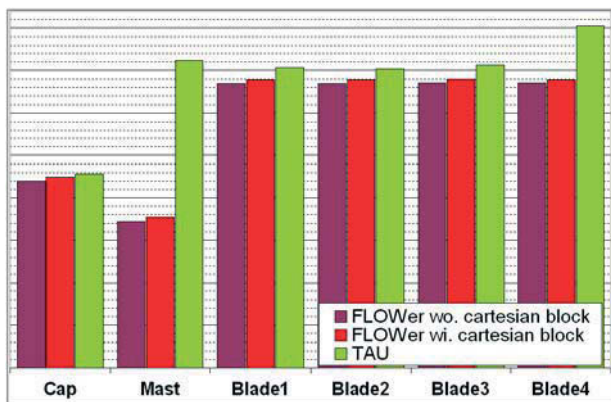


Figure 17: rotor head; drag breakdown per component

4. THE ANALYTICAL APPROACH

A flight mechanics simulation tool is one of the first to be applied when a helicopter project leaves pre-design and enters the pre-development phase. Further, it is used throughout the development and the service life of the helicopter.

Initially a flight mechanics dataset must be set-up with estimated and/or pre-computed information. It is then refined and enhanced as soon as more reliable information and test data becomes available. The present investigation describes an approach to

generate a dataset for an analytical rotor head model; a detailed presentation can be seen in [9].

4.1. Data Sources

4.1.1. CFD Results

Apart from uncertainties, e.g. as to necessary geometry simplifications, grid refinement and turbulence modelling, CFD results provide an important loads breakdown per component and a unique insight into local aerodynamics. The following CFD analyses have been resorted to:

- A quasi-steady simulation of the EC145 without rotor head in forward flight at Mach number $M = 0.21$ ([7]). Both rotors were modelled by steady actuator discs, appropriate boundary conditions were applied at engine inlet and outlet.
- A quasi-steady simulation of the EC135 without rotor head in forward flight at Mach number $M = 0.22$. The rotors were modelled through steady actuator discs, appropriate boundary conditions were applied at engine inlet and outlet.
- An unsteady simulation of the EC145 with rotating rotor head in forward flight at Mach number $M = 0.21$; neither the rotors nor the engine have been modelled.
- The FLOWer unsteady simulation of the case presented in section 3.

4.1.2. Wind Tunnel Tests

As well as CFD, wind tunnel testing is subject to a series of simplifications.

- Most of the time, the engine through-flow cannot be simulated and engine intake and exhaust have to be cowled; and none of both rotors is accounted for. Accordingly, flow interactions due to engine plume and rotor downwash are not represented.
- Since only few wind tunnels allowing full-scale testing exist, empirical data on fuselages and rotor heads is usually collected on models at scales ranging from 1/8 to 1/5. This makes then consideration of scaling effects necessary.
- Another source of discrepancy with real flight cases is reported in [10], where the level of geometrical fidelity of small scale wind tunnel models is addressed. In such cases, scaled up measured drag data may result in smaller values than for a genuine scale 1 model.

The following wind tunnel campaigns have been used:

- The campaign described in section 2.
- Wind tunnel test performed at the EC facility with

a 1/7.126 scale model of helicopter configurations similar to the current production EC135. Flow velocity of 40 m/s; polars in incidence and side-slip; and with rotating rotor head.

- Wind tunnel test performed at the EC facility with a 1/7.333 scale model of a helicopter similar to the current production EC145. Flow velocity of 40 m/s; polars in incidence and side-slip; and with rotating rotor head.
- Three different rotor heads at scales ranging from 1/7 to 1/5 were examined in a wind tunnel test performed at the University of Munich on a partial model of the BO108 [8]. This includes a rotor head of FVW design, a predecessor of the 5-bladed rotor found on the EC135. Flow velocities between 15 m/s and 30 m/s; polars in incidence and side-slip; and with rotating rotor head.

4.2. Modelling

The implementation of a rotor head model has been carried out in two steps. First, a non-rotating aerodynamics model has been placed at the rotor head centre. The verification of forces and moments was done with polars from a recent wind tunnel test. Next, refining the first model, two sets of rotating elements have been implemented, representing the pitching and the non-pitching parts of the blade stub ([9]).

The modelling approach is based on the idea that a rotor head assembly can be decomposed into a set of basic shapes, for which aerodynamic forces and moments can be calculated.

In doing so, a fundamental assumption is made: namely there is no interference between the individual elements. While this is certainly not true in reality, this reduces the modelling complexity to an acceptable level and permits an implementation with reasonable effort. The presence of the helicopter upper surface is modelled by modified incident velocities imposed on the different elements.

The main components building the rotor head assembly are exemplarily shown in Figure 18. For the current approach, the rotor head is subdivided into three groups of elements.

- Rotating and pitching elements:
The portion of the blade stub that is subject to control inputs from the swash plate is represented by a blade element model. On that portion, the geometry is subdivided into radial sections, which follow the blade pitching motion. Two-dimensional flow along the rotation movement is assumed over the individual sections.

- Rotating but non-pitching elements:
Rotating elements, not affected by blade pitch inputs, are represented by other parts of the whole model. These are the blade roots, the control rods and the swash plate attachments.
- Fixed elements:
The central part of the rotor head is composed of a set of rotational symmetrical components. The basic shapes are coaxial cylinders for the representation of the rotor mast, the hub and the swash plate. The hub cap itself is modelled as a paraboloidal shape.

Corrections are included for rotating tangential surfaces and rotating control rods, as their contribution to radial forces is not covered by the standard blade element representation.

The basic shapes used to approximate the individual components of the rotor head are circular and elliptical cylinders, rectangular blocks and rotational paraboloids. As an illustration, Figure 18 depicts the approximation of the head of an advanced 4-bladed rotor, where it appears that the rather complex geometry of the entire rotor head leads to a large number of basic shapes.



Figure 18: representation of the simplified rotor head

For each of them, the aerodynamic forces and moments are calculated and subsequently summed up over the whole rotor head. Comprehensive data on aerodynamic loads of simple shaped objects are quite seldom. The reports issued by the Engineering Sciences Data Unit (ESDU) ([2]) have here proven practical. Therein, mean fluid forces on simple objects are described as functions of geometric parameters and flow properties.

If necessary, aerodynamic data on other basic shapes can of course be supplemented. However, those listed above turned out to be sufficient for the representation of the rotor heads covered in this work.

4.3. Verification

Both rotor heads implemented using the present procedure and used for verification are the ones mounted on the EC145 and on the EC135 helicopters. Moreover the wind tunnel model (Figure 2) of the EC145 rotor head has been also implemented, which provides a test case free of scaling effects and allows comparing with wind

tunnel data.

The actual drag force, $D = q \cdot C_D \cdot S$, depends on three factors: the dynamic pressure $q = \frac{1}{2} \rho V^2$ (ρ and V denoting the density and velocity of the incident flow); the shape of the rotor head, expressed by C_D ; and finally on its size, reflected in a reference area S . The latter two defining the rotor head geometry. An analysis in terms of the key parameter $C_D \cdot S$ (drag area) allows grasping the whole geometric part (shape and size effects) at once.

The same principle applies of course to the other five aerodynamic loads. However, due to the importance of drag for the helicopter performance prediction, the following verification concentrates on the drag area.

4.3.1. Oncoming Turbulence Intensity

As described in [9], the turbulence intensity of the oncoming flow I_U has a considerable influence on the drag generated by the rotor head elements. Therefore a short analysis has been carried out to estimate the turbulence intensity that should be used for the actual verification trials.

At low I_U levels, certain rotor head elements stand in subcritical flow conditions. This is associated with higher C_D values and thus higher resulting drag forces. As the effective Reynolds number increases with higher I_U levels, the rotor head elements formerly in subcritical flow experience a supercritical flow regime, thus causing a rapid drop in overall drag. Beyond a certain I_U level, almost all parts stand in supercritical flow. Consequently, drag forces are held at lower levels and exhibit only slight variations with respect to the effective Reynolds number.

The influence of varying I_U on the different geometries is shown in Figure 19. All points have been calculated with the same reference conditions and a fuselage pitch attitude of $\alpha_{FUSE} = 0^\circ$. Compared to the EC145 rotor head, the EC135 rotor head shows less sensitivity to I_U variations. The larger diameter of the EC135 blade cuff leads to higher Reynolds numbers, thus naturally favouring supercritical flow.

The contrary can be found for the wind tunnel rotor head model (results have been scaled up for the sake of the comparison). Shorter dimensions and a lower free-stream velocity cause a stronger influence of I_U due to considerably lower Reynolds numbers. In contrast to the other two cases, the predicted drag area continues to fall with increasing I_U , though only very slowly.

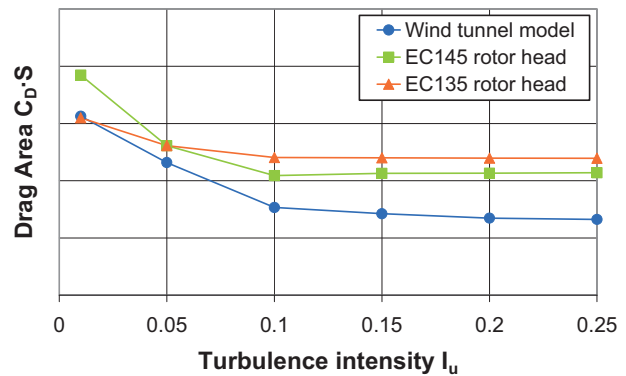


Figure 19: drag area vs. turbulence intensity

A well established turbulent flow with $I_U = 0.15$ has been assumed for the rest of the study, which has been drawn upon the following arguments: first, high turbulence levels in the main rotor wake seem likely to be expected; second, it has been observed in [9] that experimental rotor head drag is essentially independent from Reynolds number effects; and third, based on experimental data presented in [3], I_U values of this kind can tentatively be expected at the rotor head.

4.3.2. Cross-Checking

The polar in incidence of the wind tunnel campaign has been simulated with an analytical model of the small scale rotor head, thus avoiding scaling uncertainties and emulating the test set-up as closely as possible. A plot of the predicted rotor head drag area can be seen on Figure 20, where a good agreement with the experimental data can be noticed.

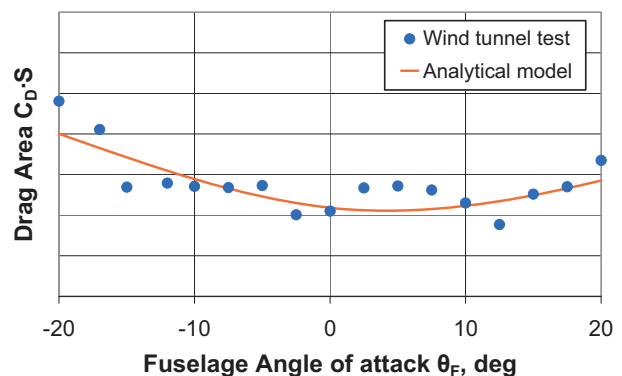


Figure 20: wind tunnel rotor head; polar in incidence

In regard of the spreading of wind tunnel points, the analytical results using the present modelling prove to provide accurate predictions. The order of magnitude as well as the trend is well matched, as is the minimum drag at around $\alpha_{FUSE} = +5^\circ$. This minimum coincides with a rotor head attitude of 0° , since the rotor mast is installed at -5° forward.

5. CONCLUSION

The different approaches to measuring, analysing, investigating, computing the aerodynamics of the rotor head, presently in use at ECD, have been presented: wind tunnel tests, CFD and analytical numerical simulations.

An analytical geometric modelling of rotor heads has been developed, where the geometry components are subdivided into elements, which in turn are modelled by basic shapes. Loads on these elemental shapes are then calculated and integrated over the whole geometry. The verification exercise has been carried out with wind tunnel data and proved very encouraging.

Based on CAD surface files, the CFD approach (FLOWer and TAU) has been applied to an isolated 5-bladed rotor head in high speed forward flight and to a wind tunnel model of the EC145 helicopter equipped with a rotating 4-bladed rotor head. Doing so, not only the surface and field solutions can be analysed in details, but also the loads distribution over body components. Validation has been done on the wind tunnel data through a comparison of numerically predicted and measured loads.

As to design optimisation tasks, the application of CFD appears particularly well suited, since this approach is directly based on CAD surfaces: every geometry change, be it only slight, is directly reflected in the computational results. The experience gained within mainly the present work but also other accompanying tasks allows drawing the following first efficient methodology:

- As long as only drag is concerned, steady state computations prove sufficient and they allow retaining the full geometric complexity of all parts of the hub, blade attachments and stubs. This can be easily handled with the unstructured approach (Centaur/TAU) using a fixed rotor head.
- On the contrary, the lift component of the aerodynamic force makes it necessary to consider rotation. Indeed modern rotor head are so designed that even blade stubs are enough streamlined so as to generate lift and hence downwash. Proper capturing of this downwash, which influences the effective incidence of the rotor head, is crucial for an accurate prediction. This in turn is most effectively dealt with the structured approach (ICEMCFD-Hexa/FLOWer) simulating both the rotation and the pitching motion controlled by the swash-plate. For the sake of even more efficiency, non-lifting parts can be geometrically simplified so as to ease not only the mesh generation process but also to further reduce computation times.

6. ACKNOWLEDGEMENTS

The wind tunnel measurement campaign at the University of Munich and the CFD part of the present work have been carried out in the frame of the German national research project SHANEL-L under grant number 20A0603C.

7. REFERENCES

- [1] Wind tunnel A data at the University of Munich: <http://www.aer.mw.tum.de/windkanal/A.de.php>
- [2] Anon.: Fluid forces on non-streamline bodies - Background notes and description of the flow phenomena. In: Engineering Sciences Data Unit, Report no. 71012, 1971.
- [3] Berry, J.D.: Unsteady Velocity Measurements Taken Behind a Model Helicopter Rotor Hub in Forward Flight. National Aeronautics and Space Administration, NASA-TM-4738, 1997.
- [4] Borie, S.; Mosca, J.; Sudre, L.; Benoit, C.; Péron, S.: Influence of Rotor Wakes On Helicopter Aerodynamic Behaviour. In: Proceedings of the 35th European Rotorcraft Forum, Hamburg, Germany, September 22nd-25th, 2009.
- [5] Gerhold, T.; Galle, M.; Friedrich, O.; Evans, J.: Calculation of Complex Three-Dimensional Configurations Employing The DLR TAU-Code. AIAA-97-0167, 1997.
- [6] Kroll, N.; Rossow, C.-C.; Becker, K.; Thiele, F.: The MEGAFLOW Project. In: Aerospace Science and Technology, Vol. 4, 2002, pp. 223-237
- [7] Le Chuiton, F.: Quasi-Steady Simulation of a Complete EC-145 Helicopter: Fuselage + Main/Tail Actuator Discs + Engines. In: Proceedings of the 31st European Rotorcraft Forum, Florence, Italy, September 13th-15th, 2005.
- [8] Meier, G.; Massinger, W.: Windkanalmessung und aerodynamische Optimierung des FEL- und FVW-Rotorkopfes sowie Vergleich mit dem BO105-Rotorkopf. Diploma Thesis from the University of Munich, 1988.
- [9] Radler, S.: Modelling of Aerodynamic Rotor Hub Forces For Helicopter Flight Mechanics Simulation. Diploma Thesis from the University of Munich, June 2009.
- [10] Sheehy, T.W.: A General Review of Helicopter Rotor Hub Drag Data. Journal of the American

Helicopter Society, No. 2, April 1977.

- [11] Schwamborn, D.; Gerhold, T.; Heinrich R.: The DLR Tau-code: Recent Applications In Research And Industry. In: Proceeding of ECCOMAS CFD 2006, Egmond aan Zee, Netherlands, September 5th-8th, 2006.
- [12] Schwarz, T.: The Overlapping Grid Technique For The Time Accurate Simulation of Rotorcraft Flows. Proceedings of the 31st European Rotorcraft Forum, September 2005
- [13] Vogel, F.; Breitsamter, C.; Adams, N.A.: Unsteady Effects of the Separated Flow at the Tail Section of a Helicopter Fuselage. In: Proceedings of the 35th European Rotorcraft Forum, Hamburg, Germany, September 22nd-25th, 2009.