Aerodynamic analysis of potential use of flow control devices on helicopter rotor blades

F Tejero, P Doerffer and O Szulc
Transonic Flows and Numerical Methods Department, The Szewalski Institute of Fluid-Flow Machinery (IMP PAN), 14 Fiszera Street, Gdańsk 80-231, Poland

E-mail: fernando.tejero@imp.gda.pl

Abstract. The interest in the application of flow control devices has been rising in the last years. Recently, several passive streamwise vortex generators have been analysed in a configuration of a curved wall nozzle within the framework of the UFAST project (Unsteady Effects of Shock Wave Induced Separation, 2005 - 2009). Experimental and numerical results proved that the technology is effective in delaying flow separation. The numerical investigation has been extended to helicopter rotor blades in hover and forward flight applying the FLOWer solver (RANS approach) implementing the chimera overlapping grids technique and high performance computing. CFD results for hover conditions confirm that the proposed passive control method reduces the flow separation increasing the thrust over power consumption. The paper presents the numerical validation for both states of flight and the possible implementation of RVGs on helicopter rotor blades.

1. Introduction
IMP PAN is putting a significant effort in the research focused on various flow control devices such as synthetic jets, wall perforation or vortex generators. In the last years, the own invention of a passive streamwise vortex generator (rod vortex generator - RVG [1]) has been studied experimentally [2] and numerically [3]. RVGs being submerged in the boundary layer provide the best compromise between the effectiveness in reducing the separation and the device drag. The basic flow control mechanism is based on the intensification of exchange of momentum in the direction normal to the wall. High momentum air is transferred to the low momentum region close to the surface and therefore the separation bubble is reduced. Rod vortex generators are defined by 5 parameters: diameter (ϕ), height (h), spacing (L), skew angle (α) and pitch angle (ϴ) (figure 1). The first three values are proportional to the boundary layer thickness while skew and pitch angles are optimized for inducing a strong streamwise vortex. A deployment of a single rod vortex generator induces streamwise vorticity due to the interaction between the mean flow and the rod (figure 2).

2. FLOWer solver
The present CFD investigation was carried out with the FLOWer code from DLR [4] which solves the Favre-averaged Navier-Stokes equations with various turbulence models. The ROT version of FLOWer incorporates the chimera overlapping grids technique and moving meshes facilitating the imposition of the blades motion. From the different turbulence closures implemented in the code, the two-equation, low-Reynolds k-ω turbulence model of LEA [5] (Linear Explicit Algebraic Stress model) was chosen. The numerical algorithm is based on a semi-discrete approach with a finite-
volume, central scheme of $2^{nd}$ order of accuracy for the spatial discretization. The same explicit, Runge-Kutta method (CFL = 2.5) of time integration was used for steady state numerical simulation of a hovering rotor, as for the internal iterations of the time-accurate implicit dual-time-stepping scheme of $2^{nd}$ order accuracy for forward flight conditions. For the steady state hover, the convergence criteria was based on a reduction of density residual by 6 orders of magnitude and stabilization of $C_T$ and $C_Q$.

For the forward flight simulation, the time needed for a rotation by 0.25˚ of azimuth was chosen for the time step (1440 time steps per revolution). At each physical time step a drop of density residual by 3 orders of magnitude proved to be sufficient to obtain accurate unsteady flow-field around the rotor.

![Figure 1. RVG schematic view.](image1)

![Figure 2. Isolated RVG inducing streamwise vortex.](image2)

3. Helicopter rotor blades

3.1. Caradonna-Tung model helicopter rotor in high-speed hover

A validation of the numerical set-up for high-speed transonic hover conditions is based on a comparison with the experimental data obtained by Caradonna and Tung (C-T) for a two-bladed model helicopter rotor [6]. The radius ($R$) of the rotor was 1.143 m with a chord ($c$) equal to 0.1905 m which leads to an aspect ratio of 6.0. The authors of the present paper have been dealing with this model rotor blade in the past using different physical models (unsteady, steady), numerical grids (block-structured, chimera), flow solvers (SPARC, FLOWer and NUMECA) and turbulence models (i.e. Spalart-Allmaras) with success [7-10]. For validation purposes, a transonic test case with a tip Mach number of 0.877, tip Reynolds number of $3.93 \cdot 10^6$ and a collective equal to 8˚ has been chosen.

3.1.1. Validation of the numerical model for hover (reference case). For the numerical investigation, the computational domain is formed by a cylindrical background grid and 2 blade component grids in chimera set-up. Two “hole” meshes are added to the domain which are used to blank unnecessary cells of the grid components (needed for the chimera interpolation procedure). The complete set of meshes consists of 114 blocks and 12.70 millions of control volumes. The cylindrical background grid (figure 3) of a height of 6.1 $R$ and radius equal to 4.0 $R$ ensures that the rotor blades are located, at least, 3.0 $R$ from the farfield. The blade grid (the blade surface is marked in yellow in figure 4) is of C-type in streamwise and H-type in crosswise directions. It spans from the surface 1.2 $c$ in all directions (radial and normal). The non-dimensional distance of the first layer of cells from the solid surface of the blade is below $y^+ = 3$ (sufficient for resolving the laminar part of the turbulent boundary layer).
The fully-turbulent steady simulation of the flow-field around the Caradonna-Tung model rotor overpredicts the experimental thrust coefficient ($C_{T-EXP} = 0.00473$) by 15% ($C_{T-CFD} = 0.00545$). Figure 5 presents the wake structure using an iso-surface of Q-criterion colored by vorticity magnitude revealing contracting and descending helical shape. For validation purposes, a comparison of chordwise pressure coefficient $C_p$ distributions was performed along the span of the blade ($r/R = 0.50, 0.68, 0.80, 0.89, 0.96$). As an example, the pressure distribution at $r/R = 0.89$ is presented in figure 6 (for more details see [10]).

Figure 3. Grid topology for C-T rotor. 

Figure 4. Blade component grid.

The acceleration of the flow on the suction side of the blade close to the tip leads to the appearance of strong shock waves which induce flow separation. The pressure distribution and separation bubble size (streamlines in white color) on the suction side of the blade are presented in the figure 6 as well. The spanwise extension of the separation bubble is between $r/R = 0.86$ and $r/R = 0.96$. The detachment point is located at $x/c = 0.30$ and almost constant in the radial direction. The reattachment point significantly varies with the cross-section. The most severe reverse flow appears at $r/R = 0.92$ where the separation bubble length is 15% of the chord.

Figure 5. Aerodynamic wake of the Caradonna-Tung rotor.

Figure 6. Contour map of pressure coefficient ($C_p$) and size of separation bubble.
3.1.2. Application of passive rod vortex generators (“RVG case”). For the reference case, the boundary layer thickness \( \delta \) upstream of the shock wave was approximately 0.008 \( \text{c} \) at \( x/c = 0.20 \). The rods were designed according to previous investigations [2] where it was concluded that the optimum design should keep the following relations:

\[
\phi = \delta / 4, \ h = \delta / 2, \ L = 2.5 \cdot \delta, \ \alpha = 45^\circ \text{ and } \Theta = 30^\circ .
\]

Due to large length of the separation bubble in spanwise direction, the spacing between the rods was increased to 5 \( \cdot \delta \) in order to reduce the number of volumes of the RVG component grid. Altogether 14 RVGs were placed between \( r/R = 0.86 \) and \( r/R = 0.96 \) (covering the whole separation bubble in the spanwise direction) and at \( x/c = 0.20 \) (12.5 \( \cdot \delta \) upstream of the flow separation). For the flow control case, the RVG chimera grid component is added to the project of the reference case chimera set-up. It is placed on the suction side of the blade and spans 0.2 \( \text{c} \) in radial direction towards the tip with respect to the outer rod, 0.5 \( \text{c} \) in radial direction towards the root with respect to the inner rod and 0.5 \( \text{c} \) in normal direction. The RVG grid component is presented in figure 7 while a detailed view of the rods in figure 8. The complete set of meshes for the RVG case consists of 616 blocks and 36.7 \( \cdot 10^6 \) of control volumes.

The numerical results of the flow past the rotor blade equipped with the proposed passive flow control system reveal the possibility of elimination of the separation bubble (figure 9). The figure presents a contour map of skin friction coefficient \( C_f \) and the location of the separation bubble (surrounded by a thick black line). The use of rod vortex generators leads to an increase of the skin friction and reduction of the separation area. The bubble is not completely eliminated but it is divided into smaller reverse flow areas. For this reason, the full elimination of the separation could be obtained by two different approaches: increase the diameter of the rods (intensification of the streamwise vortex strength) and/or decrease spacing between the rods (stronger influence in the spanwise direction).

![Figure 7. RVG component grid.](image)

![Figure 8. RVG surface grid.](image)

The numerical simulation provides a thrust coefficient equal to \( C_{T, CFD, RVG} = 0.00557 \). The use of RVGs on the blades of the Caradonna-Tung model helicopter rotor improves the thrust by 2.2\% with respect to the reference case. The use of rod vortex generators increases the power consumption by 1.2\% with respect to the reference case (\( C_{P, RVG} = 0.000630 \) against \( C_{P, REF} = 0.000623 \)). RVGs induce a parasitic drag which requires compensation in terms of power. Although this effect is a drawback of the proposed flow control, overall the performance of the helicopter rotor is improved (\( C_{T, REF}/C_{P, REF} = 8.75 \) for the reference case against \( C_{T, RVG}/C_{P, RVG} = 8.84 \) for the flow control case). It is important to mention that more severe conditions (higher collective or rotational speed) would induce stronger flow separation and the flow control system would be more effective [11].
3.2. AH-1G helicopter rotor in forward flight

A validation of the numerical set-up for forward flight conditions is based on a comparison with flight test data obtained by Cross and Watts for the AH-1G Cobra helicopter in 1988 [12]. The Tip Aerodynamics and Acoustics Test (TAAT) used highly instrumented rotor blades allowing the measurement of airloads. The TAAT report provides the measured motion of the blades (first harmonics), total thrust coefficient, pressure and normal force coefficients for different cross-sections and azimuthal positions. This data is extensively used within the helicopter community for validation of CFD codes applied to rotorcraft problems as well.

The AH-1G rotor is a two-bladed, rectangular-planform, teetering rotor with symmetrical airfoil. The blade has a linear twist of -10° and a radius of 6.7 m with a chord equal to 0.68 m which leads to an aspect ratio of 9.8. For validation purposes, the numerical results obtained for a low-, medium- and high-speed conditions have been compared against the flight test data. For all cases, the tip Mach number was approximately 0.65 while the forward flight Mach number (M∞) was varying from M∞ = 0.12 (advance ratio of 0.19) for the low-speed (152 km/h), through M∞ = 0.18 (advance ratio equal to 0.27) for the medium-speed (215 km/h), to M∞ = 0.24 (advance ratio of 0.38) for the high-speed (295 km/h). During the flight tests the instantaneous values of the rotor control angles were recorded (pitching, flapping and shaft angles) and are applied in the simulation. In contrary to the low-speed data, the presented CFD results for the medium- and high-speed forward flight conditions are unique in terms of the literature survey.

3.2.1. Validation of the numerical model for forward flight (reference case).

As first approach, the main rotor blades were isolated neglecting the influence of the fuselage, tail rotor, elasticity of the blades and no rotor trimming has been applied. Still, the remaining task was computationally very demanding. The chimera set-up used for the forward flight computation is very similar to the hover case described in section 3.1. It is formed by a background grid (in this case Cartesian) and two blade component grids. The Cartesian background grid is designed as a cuboid with dimensions of 16.4 R × 18.2 R × 18.2 R, consequently the far-field is located at least 8.0 R away from the rotor in every direction. The rotor blades are enclosed in an uniform Cartesian grid (0.12 c × 0.12 c × 0.12 c) which resolves accurately the wake behind the rotor and also supports the propagation of the low frequency acoustic pressure waves [13]. The blade grid topology is the same as for the hover case. The complete set of meshes consists of 114 blocks and 17.3·10^6 of control volumes. The numerical simulations reveal complex structures in the wake of the rotor. As an example, the Q-criterion (colored by the vorticity magnitude) visualization of the flow-field of the rotor in high-speed forward flight is presented in figure 10.
The rotor disk loading is presented in figures 11, 13 and 15 for the low-, medium- and high-speed flight cases respectively. For example in high-speed the majority of the lift is generated by fore and aft parts of the rotor disk with limited contribution from the advancing blade close to 90° and the retreating blade close to 270°. The area of reverse flow due to negative inflow velocity is growing with increasing forward flight velocity at the retreating side while the tip of the advancing blade starts to generate negative lift. The normal force distributions over the rotor disk are in line with numerical results of other researchers simulating forward flight of helicopter rotors. For the validation purposes, a normal force coefficient ($C_n$) versus azimuth ($\psi$) is compared with the flight test data at $r/R=0.86$ in figures 12, 14 and 16. For the low-speed flight case, the agreement with the recorded data is satisfactory. All main features of the measured $C_n$ distribution (i.e. local maximum of $C_n$ at the advancing side, $\psi = 90°$) are reproduced by CFD. The calculated averaged thrust coefficient is overpredicted by 2% ($C_T^{\text{EXP}} = 0.00464$ vs $C_T^{\text{CFD}} = 0.00473$). Although more discrepancies appear for the medium-speed case, the overall agreement is still acceptable. CFD reproduces fairly enough the $C_n$ values in the advancing and retreating sides, while overestimates them at $\psi = 0°$ and $\psi = 180°$ which leads to an overprediction of the thrust coefficient with respect to flight test data by 9.3% ($C_T^{\text{EXP}} = 0.00464$ vs $C_T^{\text{CFD}} = 0.00507$). Lastly, the numerical simulation of the complex flow around the AH-1G helicopter rotor blades in high-speed forward flight overpredicts the thrust coefficient by 20% ($C_T^{\text{EXP}} = 0.00474$ vs $C_T^{\text{CFD}} = 0.00569$). It is worth to mention that very similar deviations [13] were obtained in high-speed conditions for the 4-bladed rotor of the PZL W-3A “Sokół” (Falcon) helicopter.

For an additional validation for the high-speed case, the chordwise pressure coefficient $C_p$ distributions were compared against the flight test data for all recorded cross-sections and azimuthal positions. Two examples are presented in figure 10. The current computation predicts correctly the shock location at the advancing side ($\psi = 90°$, $r/R = 0.86$) and the pressure distribution at the retreating side ($\psi = 270°$, $r/R = 0.86$). The detailed 3d data was recorded for one rotor revolution (1.2 TB) in high-speed flight allowing exceptional insight into the flow complexity. Numerical results reveal the

Figure 10: AH-1G helicopter rotor wake and pressure coefficient distributions in high-speed.
appearance of flow separation due to shock wave – boundary layer interaction on the advancing side and dynamic stall on the retreating side of the rotor. Therefore, this case is convenient for the application of flow control devices (i.e. rod vortex generators).

Figure 11. Disk loading for low-speed.

Figure 12. $C_n$ comparison for low-speed.

Figure 13. Disk loading for medium-speed.

Figure 14. $C_n$ comparison for medium-speed.

Figure 15. Disk loading for high-speed.

Figure 16. $C_n$ comparison for high-speed.
4. Conclusions
The paper presents the details of the numerical simulations of helicopter rotor blades in hover and forward flight. The numerical model was validated for the C-T rotor blades in hover conditions (reference case). The appearance of flow separation suggested the possibility of application of rod vortex generators for suppression of reverse flow and improvement of the aerodynamic performance of the rotor. The numerical investigation related to the implementation of RVGs in the C-T rotor blades in high-speed hover conditions (“RVG case”) confirmed that the proposed passive flow control system enhances the thrust over power consumption. The numerical model of the AH-1G helicopter rotor in low-, medium- and high-speed forward flight was validated against the flight test data. The numerical simulation for the high-speed case revealed shock wave – boundary layer interaction on the advancing side and dynamic stall on the retreating side of the rotor – a perfect candidate for application of flow control devices designed to improve the aerodynamic performance of the rotor.

5. References
[1] Doerffer P, Flaszyński P and Szwaba R 2009 Polish patent P.389685
[2] Doerffer P 2009 UFAST Experiments Data Bank: Unsteady Effects of Shock Wave Induced Separation *IMP PAN Publishing*
[3] Flaszyński P and Tejero E F L 2013 RANS numerical simulation of effectiveness of vortex generators in a curved wall nozzle *IMP PAN Report No. 366/2013*
[4] Rosow C C, Kroll N and Schwamborn D 2006 The MEGAFLOW project – numerical flow simulation for aircraft *Progress in Industrial Mathematics at ECMI 2004* 8 pp 3-33
[5] Rung T, Lübcke H, Franke M, Xue, L, Thiele F and Fu S 1999 Assessment of explicit algebraic stress models in transonic flows *Proc. of the 4th Int. Symposium on Engineering Turbulence Modelling and Measurements* (Ajaccio)
[6] Caradonna F X, Tung C 1981 Experimental and analytical studies of a model helicopter rotor in hover *NASA Technical Memorandum 81232*
[7] Doerffer P and Szulc O 2008 Numerical simulation of model helicopter rotor in hover *TASK Quarterly* 12 3
[8] Doerffer P, Szulc O, Tejero E F L and Martinez S J 2014 Aerodynamic and aero-acoustic analysis of helicopter rotor blades in hover *Bubak, M., (ed.), E-science on Distributed Computing Infrastructure: PLGrid Plus, LNCS, Springer*
[9] Doerffer P, Tejero E F L and Szulc O 2014 Numerical simulation of model helicopter rotor in hover using chimera overlapping grids technique *IMP PAN Report No. 29/2014*
[10] Tejero E F L, Doerffer P, Flaszyński P and Szulc O 2014 Numerical investigation of rod vortex generators on hovering helicopter rotor blade *Proc. of the 6th European Conf. on Computational Fluid Dynamics* (Barcelona)
[11] Tejero E F L, Doerffer P and Szulc O 2013 Effect of passive air jet vortex generator on NACA0012 performance *Proc. of the 5th European Conf. for Aerospace Sciences* (Munich)
[12] Cross J L and Watts M E 1988 Tip aerodynamics and acoustics test *NASA Reference Publication 1179*
[13] Doerffer P, Szulc O, Tejero E F L, Żółtak J and Malecki J 2014 Numerical simulation of high-speed impulsive noise of the PZLW-3A “Sokół” (Falcon) helicopter main rotor in forward flight *Proc. of the 6th European Conf. on Computational Fluid Dynamics* (Barcelona)

Acknowledgments
This work was supported by the 7th Framework Programme projects IMESCON (Grant Agreement PITN-GA-2010-264672) and STADYWICO (FP7/2007-2013 under grant agreement no. 251309). The authors would like to thank the Academic Supercomputing Center TASK, Gdańsk (Poland) and PL-Grid Infrastructure for providing excellent HPC resources.