

Assessment of simulation techniques for rotor loads prediction

R. Steijl*, G. Barakos, K. Badcock

CFD Laboratory, Department of Aerospace Engineering, University of Glasgow, Glasgow G12 8QQ, UK

Abstract

Numerical simulation of hovering rotor cases has been undertaken using computational fluid dynamics (CFD) as well as reduced aerodynamic models in an attempt to identify the advantages and disadvantages of each method. A well-known test case has been identified corresponding to the experiments of Caradonna and Tung on a Mach scaled rotor. On the CFD front, the Euler's equations have been used in conjunction with the control volume method. The reduced model was based on the indicial approach and coupled with a prescribed wake model. The obtained results indicate that prediction of the rotor loads is possible using CFD despite the lack of resolution in the wake region. Grids of up to 3 million points per blade were used on average, each calculation required about 12 hours on a 10-node Beowulf cluster. Although the prescribed wake model was much faster than CFD, it was found to be sensitive to the 'tuning' parameters of the wake and consequently its range of application is restricted. Present results indicate that CFD methods can reliably predict the loads of the rotor but the lack of resolution at the wake restricts their routine application to the rotor design task. Problems like rotor/fuselage or main-rotor/tail-rotor interaction are still too expensive to be computed routinely.

Keywords: Hovering rotors; Rotorcraft CFD; Rotor wake; Indicial methods; Rotor loads

1. Introduction

In current practice, the design of helicopter rotors is based on reduced aerodynamic models while the use of computational fluid dynamics (CFD) by the rotorcraft industry still remains limited. This is mainly due to the highly complex flow field encountered even in the simplest of rotor flows like hover. Strong vortices interacting with each other and the rotor blades, formation of a complex spiral wake behind the rotor, transition to turbulence and wide variation of the Mach and Reynolds numbers around the azimuth are a few of the difficult issues CFD methods have to cope with. On the other hand, reduced models in use by the industry rely on prescribed or free wake methods to predict the induced flow on the rotor blades and eventually the performance of the rotor system. Without the wake, such methods have no way of accounting for the flow configuration near the blades and consequently their performance is directly related to the realism of the employed wake model. Even for a hovering rotor, where the wake structure is to a great extent known, several

parameters, like the contraction ratio and the wake aging, need to be adjusted. The availability of experimental data makes this process straightforward for existing rotor designs but it may put severe restrictions on the applicability of the reduced model in the design of new rotors.

The wake consists of a vortex sheet emanating from the trailing edge of the blades and two tip vortices from each blade edge. This wake spirals down below the rotor. Compared to the induced vortex wake of an aircraft or a wing, the vortex wake of a rotor remains in close proximity of the rotor for a much longer duration. The presence of this complex wake has a strong effect on the flow around the rotor blades. It is commonly argued that 3–4 revolutions of the spiralling vortex wake need to be resolved for a proper simulation of the flow field of the rotor in hover conditions. Even with modern parallel computing capabilities this remains a very difficult task, and typically CFD studies of rotors using the Euler or Navier-Stokes equations do not preserve the vortex wake beyond one revolution or less. However, the computed loads on the rotor can be in very good agreement with experimental data.

In view of the above, the objective of this work is to compare CFD results for hovering rotors with the

* Corresponding author. Tel.: +44 (141) 330 8646; Fax: +44 (141) 330 5560; E-mail: rsteijl@eng.gla.ac.uk

output of a reduced model and assess the performance of the two methods. The 2-bladed model rotor from the experimental work of Caradonna et al. [1] is used as a test case. For this test case, the tip Mach number was 0.44 and the collective 8° . Experimental data for the surface pressure are available for 5 rotor blade sections: at 0.5, 0.68, 0.8, 0.89 and 0.96 of the rotor radius.

2. Numerical methods

2.1. CFD method

The detailed description of the employed CFD solver can be found in [2], only the hover-specific features are discussed here. The rotor hover problem is simulated here as a steady flow problem in a non-inertial frame of reference. A fixed curvilinear coordinate system is used, as described by Chen et al. [3]. A combination of a source term for the three momentum conservation equations and the use of a mesh velocity in the formulation of the 3D Euler equations in the fixed curvilinear coordinate system represent the centripetal and Coriolis acceleration terms that result from the rotor rotation. In the non-inertial frame of reference, the Euler equations become:

$$\frac{\partial \vec{w}}{\partial t} + \frac{\partial \vec{E}}{\partial \xi} + \frac{\partial \vec{F}}{\partial \eta} + \frac{\partial \vec{G}}{\partial \zeta} = \vec{R} \quad (1)$$

where $\vec{w} = [\rho, \rho u, \rho v, \rho w, \rho e]^T$ is the vector of conserved variables. \vec{E} , \vec{F} and \vec{G} represent the usual inviscid fluxes. However, the contra-variant velocities and source term are now:

$$[U, V, W]^T = [u, v, w]^T - (\vec{\omega} \times \vec{r}) ;$$

$$\vec{R} = \frac{1}{J} [0, -\omega \times \vec{u}]^T \quad (2)$$

where ω is the rotation vector of the rotor and \vec{r} the position vector of the point considered.

Two different approaches to impose far-field boundary conditions are used and compared. The first approach imposes unperturbed free-stream conditions at the far-field of the domain and linear extrapolation in the vertical direction, on the inflow and outflow boundaries. The second approach is termed a ‘potential sink’ or ‘Froude boundary condition’. In this case, a potential sink is placed at the rotor origin, as used by [4,5,6] and others. Furthermore, based on actuator-disk theory, a constant axial (outflow) velocity is prescribed on a circular part of the outflow boundary face. The magnitude of this velocity is determined by the rotor thrust (which gives the induced axial velocity through the rotor disk according to actuator-disk theory) and the

outflow radius, for which an empirical relation (described in [6]) is used that approximates the wake contraction to $R/\sqrt{2}$ far from the disk. On the remainder of the far-field boundary, the velocity due to the potential sink is imposed. The strength of the sink is chosen to balance the mass flows into and out of the computational domain.

For the present series of simulations for the 2-bladed rotor, computational meshes with periodic conditions on the symmetry plane were used, reducing by half the size of the problem.

2.2. Reduced model

The reduced aerodynamic model employed for this work is based on the aerodynamic model developed by Beddoes et al. [7]. This model is aimed at calculating the unsteady aerodynamic forces encountered in helicopter rotor operating environments. During the development, several criteria including simplicity to allow for quick computational times, incorporation of both attached flow conditions and separated flow conditions and the ability to include arbitrary forcing functions (necessary to adequately predict the forcing encountered within the helicopter rotor environment) were considered. Details of the method used can be found in [8].

To include the effects of the wake in the reduced model, the method of wake prediction must be compatible with the indicial model for unsteady aerodynamic loading. The influence of a time-dependent shed wake is included implicitly in the indicial method, but the effect of tip vortices is included explicitly by the wake model. The wake model tracks the individual vortex elements and sums the contributions from each element to determine the effect of trailing vortices in the near wake on the blade incidence.

3. Results and discussion

Figure 1 compares the wake structure of the employed reduced model against resolved tip vortex from the CFD result. For the reduced model, a small contraction ratio has been used that matches the measurements of Caradonna et al. [1]. The predicted contraction of the CFD result is consistent with these measurements, however, the dissipation of the tip vortex due to a lack of resolution is apparent. Table 1 presents the comparison between experiments and predictive methods for the thrust coefficient of the Caradonna–Tung rotor. As can be seen, both CFD and the indicial methods provided accurate results with errors less than 5% with respect to the measured value. The sensitivity of the prescribed wake model to the wake contraction ratio is also revealed and as can be seen, a change of the contraction

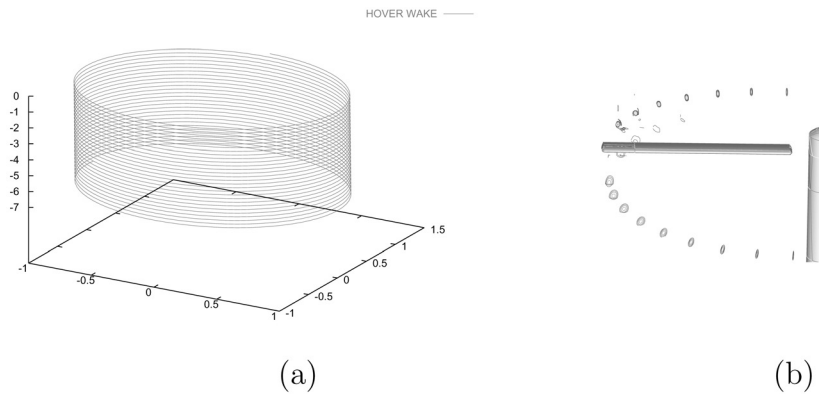


Fig. 1. Wake structures for the reduced model (a) and the CFD method (b).

Table 1

Comparison between experiments, CFD and reduced model for the thrust coefficient of the 2-bladed Caradonna–Tung rotor ($M_{tip} = 0.44$, 8° collective pitch)

| Method | Thrust coefficient |
|--|--------------------|
| Experiment | 0.00460 |
| CFD | 0.0045 |
| Indicial method | 0.004503 |
| Indicial method + 10% wake contraction | 0.004205 |
| Indicial method + 10% wake aging | 0.004381 |

ratio of 10% resulted in disproportionately higher errors in the thrust coefficient. The same can be said for the wake age parameter. A similar dependence on the contraction ratio was found by Li et al.[9]. For the present case, wake data have been provided as part of the experimental programme, that could be used in the reduced model. However, most scaled-rotor experiments are restricted to blade load measurements, making the use of the reduced model more difficult.

During the initial phases of rotor design, the most important parameter is the spanwise loading and for this reason, results are presented in Fig. 2 for the case of the Caradonna–Tung rotor. Measurements at five spanwise stations have been provided and as can be seen, CFD results compare well against measurements, for both grids with 2 million and 3 million points per blade. For the domain size used, the effect of the different far-field boundary conditions is very small. For the two grids, care has been taken to provide the refinement close to the body and obtain a good balance between the grid density on the surface of the rotor and the wake. In fact, results on grids of 1 million points (not shown here) were also found to predict the blade loads quite well. The result of the indicial method similarly compares well against measurements, when the appropriate wake

parameters are used. The sensitivity of the spanwise loading to the wake contraction and wake aging is quite strong. In contrast, the CFD results with different levels of spatial resolution of the wake do not show this strong sensitivity of the spanwise loading.

4. Conclusions

Numerical simulation of the hovering rotor problem has been undertaken using CFD and indicial methods. For the 2-bladed Caradonna–Tung rotor, the obtained results indicated that the CFD technique is very accurate in predicting the rotor loads but lacks resolution of the wake structure. The indicial method used for this work

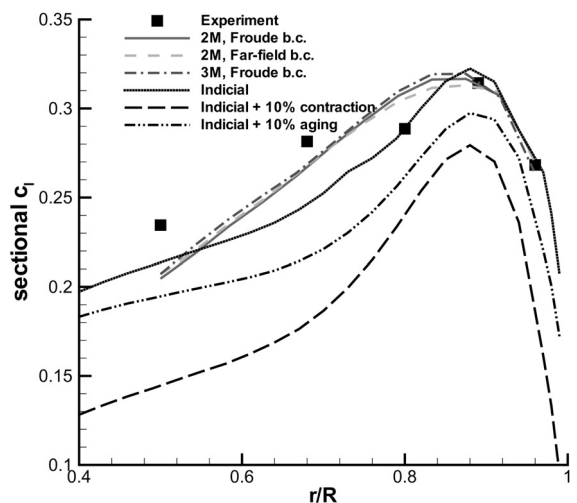


Fig. 2. Comparison between experiment and simulation for the sectional lift distribution of the Caradonna–Tung rotor ($M_{tip} = 0.44$, 8° collective pitch).

was very efficient and gave good results for the rotor loads but was found to be sensitive to the parameters of the employed free wake, making its application to different hover cases difficult. The main conclusion of this work is that CFD can be used in routine calculations of rotor performance in hover even with grids of moderate density that do not allow for the detailed resolution of the wake structure. It has to be noted that the exact structure of the rotor wake is crucial in problems like rotor/fuselage or main-rotor/tail-rotor interaction and thus better numerical schemes and computational techniques are necessary. In the future, grid adaptation and higher-order methods are to be assessed in order to counter this problem. This work is part of a wider research programme aiming at improving CFD methods for rotorcraft applications.

Acknowledgements

The financial support by the Engineering and Physical Sciences Research Council (EP-SRC) is gratefully acknowledged. This work is part of the Rotorcraft Aeromechanics Defense and Aerospace Research Partnership (DARP) programme on predictive methods for helicopter rotors.

References

- [1] Caradonna FX, Tung C. Experimental and analytical studies of a model helicopter rotor in hover. NASA Technical Memorandum 81232, 1981.
- [2] Badcock K, Richards B, Woodgate M. Elements of computational fluid dynamics on block structured grids using implicit solvers. *Progress Aerospace Sci* 2000;36(5–6):351–392.
- [3] Chen CL, McCroskey WJ, Obayashi S. Numerical solutions of forward-flight rotor flow using an upwind method. *J Aircraft* 1991;28(6):374–380.
- [4] Srinivasan GR, Baeder JD. TURNS: a free-wake Euler-Navier-Stokes numerical method for helicopter rotors. *AIAA J* 1993;31(5):959–962.
- [5] Pomin H, Wagner S. Navier-Stokes analysis of helicopter rotor aerodynamics in hover and forward flight. *J Aircraft* 2002;39(5):813–821.
- [6] Biava M, Vigeveno L. The effect of far-field boundary conditions on tip vortex path predictions in hovering. In: *Proc of CEAS Aerospace Aerodynamics Research Conference*, Cambridge, 10–13 June, 2002.
- [7] Beddoes TS, Leishman JG. A generalised model for airfoil unsteady aerodynamic behaviour and dynamic stall using the indicial method. In: *Proc of 42nd Annual Forum of the American Helicopter Society*, 1986.
- [8] Beedy J. Summary of Beddoes/WHL Aerodynamic Model. *Aerospace Eng Report* 0207, Glasgow University, 2004.
- [9] Li H, Burggraf O, Conlisk A. Formation of a rotor tip vortex. *J Aircraft* 2002;39(5):739–749.